© 2014 Sarah Berastegui-Vidalle

APPLICATION AND ENHANCEMENT OF THE SRM MODEL

BY

SARAH BERASTEGUI-VIDALLE

THESIS

Submitted in partial fulfillment of the requirements
for the degree of Master of Science in Environmental Engineering in Civil Engineering
in the Graduate College of the
University of Illinois at Urbana-Champaign, 2014

Urbana, Illinois

Adviser:

Professor Marcelo H. García

ABSTRACT

The main focus of this thesis has been to apply and to develop the storage routing model. The storage routing model is a method of hydraulic numerical calculation. It is based on the Saint-Venant equations and works under the quasi steady dynamic wave assumption. It first creates HPGs and VPGs (respectively upstream water surface elevation and storage as function of the discharge and the downstream water surface elevation) and uses the interpolations of these graphs at each step of the calculation. This model has been developed for systems composed of circular conduits. In this thesis, at first the accuracy of the model has been checked for systems with circular pipes, the Wallingford research station and some theoretical models from Matthew A. Hoys thesis [**?**]. Then the model has been adapted to river systems that include non-prismatic channels with input HPGs/VPGs and has been applied to the Belo Monte canal and the Main Stem Chicago river. The comparison with field data or with other models results is satisfying, the error (max(abs((xmodel-xdata)/xdata))) is less than 10 % in all the cases. In the last part, a study is made on how the spacing used for the numerical calculation criteria can affect the accuracy.

ACKNOWLEDGMENTS

I would first like to thank Marcelo H. García for suggesting and guiding the research presented in this thesis and for his support throughout the duration of my education at the University of Illinois. I would also like to thank Blake J. Landry for his guidance, his time, thoughtful discussions, and careful review of the work included in this thesis.

Special thanks is due to Nils Oberg, who developed the code used to generate the Hydraulic Performance Graphs and Volumetric Performance Graphs with Arturo Leon, and developed the tool to generate the HPGs and VPGs using HEC-RAS geometry with Andrew R Waratuke.

I would like to thank the numerous officemates and research assistants with whom I have had the pleasure of working with at the Ven Te Chow Hydrosystems Laboratory at the University of Illinois. Finally, I would like to thank my friends and family for their support and their encouragement.

CONTENTS

LIST OF FIGURES v

LIST OF SYMBOLS viii

Chapter 1 INTRODUCTION 1
1.1 Description and development of HPG and VPG 2
1.2 Storage Routing Method (SRM) 6

Chapter 2 SOME RESULTS WITH SRM 10
2.1 Wallingford Hydraulic Research Station 10
2.2 Theoretical Model from Matthew Hoy's Thesis 16
2.3 The Canal 22
2.4 The Chicago River Main Stem 27
2.5 Theoretical cases 34

Chapter 3 A SPACING CRITERIA 43
3.1 For steady flow (backwater length) 44
3.2 Another spacing criterion that applies for unsteady flow 47
3.3 Verification of unsteady flow spacing criteria 49

Chapter 4 CONCLUSION 58

Appendix A USER'S MANUAL 60
A.1 Introduction 60
A.2 Description of the code 62
A.3 How to use SRM 70
A.4 Alternatives for using SRM 72

LIST OF FIGURES

1.1 Example of a backwater calculation 4
1.2 Example of the division of a system 8

2.1 Inflow hydrograph for Wallingford Hydraulic Research Station 11
2.2 Water surface elevation at 28.4 ft. downstream of the entrance of the pipe 12
2.3 Water surface elevation at 255.7 ft. downstream of the entrance of the pipe 13
2.4 Water surface elevation at 483 ft. downstream of the entrance of the pipe 14
2.5 Water surface elevation at 710.2 ft. downstream of the entrance of the pipe 15
2.6 Inflow hydrograph for Theoretical Model from Matthew Hoy's Thesis 16
2.7 Water surface elevation at the entrance of the pipe 17
2.8 Water surface elevation at 2500 ft. downstream of the entrance of the pipe 18
2.9 Water surface elevation at 5000 ft. downstream of the entrance of the pipe 19
2.10 Discharge at 2500 ft. downstream of the entrance of the pipe . 20
2.11 Discharge at 5000 ft. downstream of the entrance of the pipe . 21
2.12 Inflow hydrograph for The Canal 22
2.13 Cross-section geometry of The Canal 23
2.14 Water surface elevation at the entrance of the canal 23
2.15 Water surface elevation at the exit of the canal 24
2.16 Discharge at the entrance of the canal 25
2.17 Discharge at the exit of the canal 26
2.18 View of the Chicago River Main Stem 27
2.19 Geometry of the Chicago River Main Stem 27
2.20 Geometry of the Chicago River Main Stem 28
2.21 Stage hydrograph at the upstream point of Main Stem 29
2.22 Model for the stage hydrograph at the upstream point of Main Stem 30
2.23 Discharge in the Main Stem 31

2.24 Water surface elevation in the Main Stem 32
2.25 error between SRM and HEC-RAS results 33
2.26 First Inflow hydrograph 34
2.27 Second Inflow hydrograph 35
2.28 Discharge at several station for $S = 2 \times 10^{-6}$ with the first hydrograph 35
2.29 Water surface elevation at several station for $S = 2 \times 10^{-6}$ with the first hydrograph 36
2.30 Discharge at several station for $S = 2 \times 10^{-6}$ with the second hydrograph 37
2.31 Water surface elevation at several station for $S = 2 \times 10^{-6}$ with the second hydrograph 38
2.32 Discharge at several station for $S = 2 \times 10^{-5}$ with the first hydrograph 39
2.33 Water surface elevation at several station for $S = 2 \times 10^{-5}$ with the first hydrograph 40
2.34 Discharge at several station for $S = 2 \times 10^{-5}$ with the second hydrograph 41
2.35 Water surface elevation at several station for $S = 2 \times 10^{-5}$ with the second hydrograph 42

3.1 the error vs the frequency of ratio higher than 1 50
3.2 the error vs the frequency of ratio higher than 10 51
3.3 the error vs the frequency of ratio higher than 100 52
3.4 results for the discharge with a spacing of 113.6 ft 53
3.5 results for the discharge with a spacing of 50 ft 54
3.6 results for the discharge with a spacing of 50 m 55
3.7 results for the discharge with a spacing of 6250 ft 56
3.8 distribution of the ratio α for all cases at the "limit" spacing . 57

A.1 overview of the code 62
A.2 set the global variable *interceptor* 63
A.3 The junctions in SRM 63
A.4 initialization: steady flow calculation 65
A.5 unsteady flow calculation 68
A.6 What to change i the main file *makedecisions* 70
A.7 progress of the calculation 71
A.8 HEC-RAS file for the Chicago River 73
A.9 Division of the Chicago River Main stem reach 74
A.10 Two possible directions for a reach 74
A.11 where to find the invert elevation 75
A.12 how to set the inverts file 75
A.13 How to classify the files 76
A.14 code to create the HPG files 76
A.15 How to check the files 77

A.16 the *interceptor* variable 77
A.17 the conduit *field* of the *interceptor* variable, the red fields are asbent of the conduit variable created to input the HPGs/VPGs 78
A.18 How to use the file containing HPGs/VPGs in SRM code . . . 78
A.19 the boundary conditions in the code 79
A.20 the boundary conditions in the code 79

LIST OF SYMBOLS

A	is the cross section area in m^2
B	is the channel width in $[m]$
D	is the hydraulic depth in $[m]$
F	is a force applying to the material inside the volume V in $N = kg.m/s^2$
Fr	is the Froude number
f	is an equation for the Newton-Raphson method
g	is the acceleration due to gravity in m/s^2
h	is the water surface elevation in m
h_n	is the normal water surface elevation in m
K_n	is a unit conversion coefficient. Equal to 1 for I.S. and to 1.486 for E.U.
L_b	is the backwater length in m
L_u	length of the channel affected by unsteadiness
n	is mannings roughness coefficient
Q	is the discharge in m^3/s
Q_{in}	is an discharge coming in the junction in m^3/s
Q_{out}	is an discharge going out of the junction in m^3/s
Q_n	is the discharge for a normal regime on in m^3/s

q	is the lateral inflow in m^2/s
v	is the velocity of the flow in m/s
R	is the hydraulic radius in m
S	is the surface delineating the volume considered form outside in m^2
S_0	is the slope of the bottom of the channel
S_f	is the friction slope
t	is the time in s
V	is the volume considered for derivations in m^3
v	is the velocity of the fluid entering the volume V through dS in m/s
x	is the longitudinal dimension in m
y_i	is the water surface elevation at point i in m
y_{limit}	is the water elevation of one arbitrary chosen node in a junction (a junction is several nodes with same location) in m
y_{up}	is the water surface elevation of the upstream end of a link in m
y_{down}	is the water surface elevation of the downstream end of a link in m
α	is the ration of the spacing over the unsteadiness criteria
β	is the Boussinesq coefficient
Δx	is the spacing in m
Δt	is the time step in s
η	is a small perturbation to normal in m
ρ	is the density of the material (fluid) inside the volume in kg/m^3
θ	is the angle between the channel and the horizontal (slope angle)

ζ	is a small perturbation of the discharge to the normal regime in m^3/s

Chapter 1

INTRODUCTION

Irregular precipitations as well as irregular use of water cause the flow in rivers and hydraulic infrastructure to be unsteady and non-uniform. During critical events (heavy rainstorm for example), it is useful to know some estimations of the discharge and the water surface elevation, it must helps preventing overflows and floods. A typical way to obtain these values is to use the Saint-Venant equations and solve them numerically. These equations are non-linear and quite complex. Thus is it often useful to make some assumptions to neglect some terms and simplify the calculation. It may allow simpler computation and faster results without losing to much accuracy. SRM (storage routing method) is a matlab code that applies this method under the quasi-steady dynamic wave assumption. It has been developed and enhanced by previous authors' work in order to "create a method for modeling these unsteady flows in a straightforward and efficient manner" (Matthew A. Hoy, 2005) [**?**].

The methods of calculation based on Saint-Venant apply for a one-dimensional flow, an incompressible fluid, with a hydrostatic pressure, no scour or deposition and a small slope. These equations being non-linear, no general analytical solution exists. In the case of unsteady flow, no term is neglected, the computation can be intensive. On the other hand, when too many terms are neglected, the result might not be accurate enough. In some cases, the convective acceleration can be neglected and the quasi-steady dynamic wave is a good compromise between accuracy and computation velocity. SRM is one method of calculation based on the quasi-steady dynamic wave assumption. It creates table of values of upstream water surface elevation (HPGs) and volume of water (VPGs) depending on the discharge and the downstream water surface elevation for all portions of the system considered beforehand and uses interpolations of these graphs at each time step of the calculation

to find the solution.
In the case of SRM, an aspect that can be critical for the time of computation and the accuracy is the spacing Δx taken. When it is too small, the time of computation is too long and the code might take days to give some results. When the spacing is large, the results might lack accuracy and when it is very large, the code might crash or gives meaningless results.
Previously, a spacing criterion based on the backwater length has been established for unsteady flow (Samuels, P. G. 1989) [**?**]. The same kind of derivation can be done to find the length of reach affected by unsteadiness and thus another spacing criterion. The backwater length is already taken into account in the HPG, the spacing criterion associated to this length does not have to be taken into account when dividing the system into parts. The criterion that applies for this, is the one linked to unsteadiness.

1.1 Description and development of HPG and VPG

1.1.1 Saint-Venant equations

Saint-Venant equations are a special case of the momentum (1.1a) and the conservation (1.1b) equations (Ven Te Chow et. Al., 1988, p272) [**?**].

$$0 = \frac{d}{dt} \iiint_V \rho \mathrm{d}V + \oiint_V \rho v \mathrm{d}S \tag{1.1a}$$

$$\sum F = \frac{d}{dt} \iiint_V v\rho \mathrm{d}V + \oiint_V \rho v^2 \mathrm{d}S \tag{1.1b}$$

It applies under the following assumptions:

- The flow is 1-dimensionnal, only the variations in the flow wise direction are considered.
- The pressure is assumed to be hydrostatic in the entire channel, the vertical acceleration are neglected

- The channel can be considered as a straight line
- The geometry of the channel is fixed (no effect of scour and deposition)
- The slope is small
- The fluid is incompressible

Applying the previous assumptions the momentum and the conservation equations become respectively the first (1.2a) and the second (1.2b) Saint-Venant equations:

$$\frac{\partial Q}{\partial t} + \frac{\partial \frac{\beta Q^2}{A}}{\partial x} + gA\left(\frac{\partial h}{\partial x} + S_0 + S_f\right) = 0 \tag{1.2a}$$

$$\frac{\partial Q}{\partial x} + \frac{\partial A}{\partial t} + = 0 \tag{1.2b}$$

With The Boussinesq coefficient defined by the following formula:

$$\beta = \frac{1}{v^2 A} \iint v^2 \mathrm{d}A$$

And the friction slope:

$$S_f = \frac{v^2 n^2}{{K_n}^2 R^{\frac{4}{3}}}$$

1.1.2 Gradually varied flow calculation for steady flow

In the case of steady flow, the terms in $\frac{\partial Q}{\partial t}$, $\frac{\partial Q}{\partial x}$ and $\frac{\partial A}{\partial t}$ are equal to zero. The equations of Saint-Venant become one equation:

$$\frac{\beta Q^2}{A^2}\frac{\partial A}{\partial x} + gA\left(\frac{\partial h}{\partial x} + S_0 + S_f\right) = 0$$

Which becomes a gradually varied flow equation (Ven Te Chow, 1959, p134) [**?**]:

$$\frac{dy}{dx} = \frac{S_0 - S_f}{cos\,(\theta) - {F_r}^2} \tag{1.3}$$

Where ${F_r}^2 = \frac{\beta v^2}{gD}$ is the Froude Number squared. Equation (1.2b) allows the calcualtion of the water elevation along the channel (Terry W. Sturm, 2001, p200) [**?**]. This is the backwater calculation.

Typically, y_i is a function of y_{i+1} and the discharge. The water surface elevation is calculated knowing the downstream water surface elevation and the discharge.
The value of the upstream water surface elevation can be obtained from the back water calculation:

$$y_{up} = f\,(y_{down}, Q)$$

Complementary calculation for the volume

The cross-section area is a function of the geometry of the channel and the water surface elevation. Thus, through the backwater calculation, the cross-section area is known. The volume can be calculated using this formula:

$$V = \int_{x_{down}}^{x_{up}} A \mathrm{d}x$$

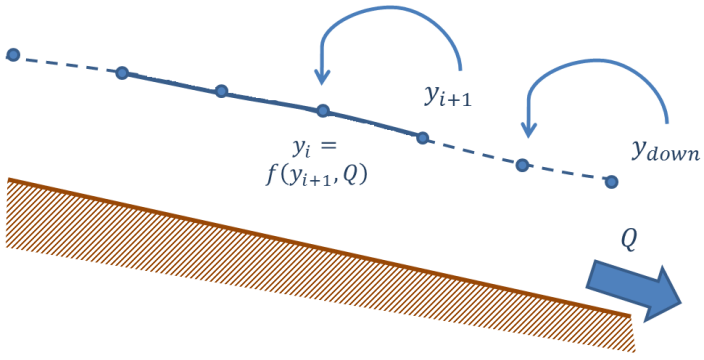


Figure 1.1: Example of a backwater calculation

Which can be discretized and used during the backwater calculation to find the storage:

$$V = \sum A\Delta x \tag{1.4}$$

Thus the volume of water in the reach is also a function of the downstream water surface elevation and the discharge:

$$V = f\left(y_{down}, Q\right)$$

1.1.3 The Hydraulic Performance Graph (HPG)

The Hydraulic Performance graph is described as (Juan A. Gonzales Castro and Ben Chie Yen, 2000, p.16) [**?**] a group of Hydraulic Performance Curves for a channel reach. An hydraulic performance Curve for a given channel reach and a given discharge is a curve giving the values of the upstream water surface elevation as a function of the downstream water surface elevation in the case of subcritical flow.
In most cases, the HPG is obtained using a gradually flow calculation. Sometimes it can be obtained through other method of calculation. In the case of pipes, it can be extended to include pressurized flow.

1.1.4 The Volumetric Performance Graph (VPG)

The volumetric Performance Graph is similar to the HPG (Matthew A. Hoy, 2005) [**?**]; however it gives the volume of water stored in the river reach or conduit as a function of the discharge and the downstream water surface elevation for subcritical flow, instead of the upstream water surface elevation. It is a group of Volumetric Performance Curves for a channel reach. A volumetric performance Curve for a given channel reach and a given discharge is a curve giving the values of the volume of water as a function of the

downstream water surface elevation in the case of subcritical flow.
The VPG can be obtained with the same method as the HPG and can also be extended to include pressurized flow.

1.1.5 HPG and VPG extension to pressurized flow

The method of backwater calculation obtained with the Saint-Venant equations works only for non pressurized flow. In the case of pressurized flow, the upstream water surface elevation and the volume of water can be calculated with another method.
To extend the HPG to pressurized flow, the method developed by Zimmer, A. et al. (2013, p937) [**?**] was to use Darcy-Weisbach equations (Terry W. Sturm, 2001, p110) [**?**] :

$$\frac{h_f}{L} = \frac{f}{4R}\frac{v^2}{2g}$$

Where f can be expressed as:

$$f = \frac{8gn^2}{{K_n}^2 R^{\frac{1}{3}}}$$

The friction loss h_f gives the difference between the upstream and the downstream water surface elevation as a function of the discharge.

Therefore, the upstream water surface elevation is given as a function of the downstream water surface elevation and the discharge and the volume can be calculated with the same method as seen previously (1.4).

1.2 Storage Routing Method (SRM)

For a given geometry of a river or a pipe network and two boundary conditions (In the case of one reach, two boundary conditions are necessary. It can be

two inflow, outflow or stage hydrographs, or a rating curve), an unsteady flow calculation can be done. SRM is one method of calculation of unsteady flow based on the Saint-Venant equations under the quasi-steady dynamic wave assumption. It interpolates tables of values of discharges, water surfaces elevations and volumes of water for each component of the system. In the case of conduits/reaches, these tables are HPGs/VPGs and are extended to pressurized flow when necessary (Zimmer, A. et. al., 2013) [**?**].

1.2.1 How it works

At the first time step, the calculation is done assuming a steady flow, for the next time steps, the storage calculated at the previous time step is used to do the calculation for unsteady flow. For each time step, a first guess is given for the values of water surface elevation and discharge at each node and it uses Newton-Raphson to make it converge to the solution.
To solve the equations, the system is divided into nodes and links (Zimmer, A. et al., 2013, p938-939) [**?**]. At each node, there are two unknowns: the discharge and the water surface elevation.
The pipes or portions of pipes, the river reaches, the pumps, the storage units and other features are considered as links. Each link gives two equations, one is linked to the conservation of mass, the other one to the momentum equation.
A junction is where there are more than two links or where there are two links and an inflow. At each junction, ghost nodes are created correspond to every links' end and every inflow in order to facilitate the numerical resolution. Each junctions gives as many equations as it has nodes.

Therefore, this becomes a system of unknowns and equations.
There are usually more unknowns than equations. To complete the number of equations, boundary conditions are necessary.

The equations are (Zimmer, A. et al., 2013, p947) [**?**]:

$$F = (f_i)$$

- for junctions:

$$\begin{cases} f = y_i - y_j & \text{if } i \neq j \\ f = \sum Q_{in} - \sum Q_{out} \end{cases}$$

- for links:

$$\begin{cases} f = \Delta t \left(Q_{in} - Q_{out}\right) - \Delta volume & \text{conservation equation} \\ f = y_{up} - f\left(y_{down}, Q\right) & \text{momentum equation} \end{cases}$$

- Special case of conduits/reaches

$$\begin{cases} f = \Delta t \left(Q_{in} - Q_{out}\right) - \Delta VPG\left(\frac{Q_{in}+Q_{out}}{2}\right) \\ f = y_{up} - HPG\left(y_{down}, Q\right) \end{cases}$$

- The boundary conditions

$$f = \begin{cases} Q - inflow/outflow & \text{inflow or outflow hydrograph} \\ f = y - stage & \text{stage hydrograph} \\ f = rating\left(y, Q\right) & \text{rating curve} \end{cases}$$

The method used to find the roots of this system is Newton Raphson. The partial derivatives of these functions with respect to the discharges and the water surface elevations at every node need to be calculated:

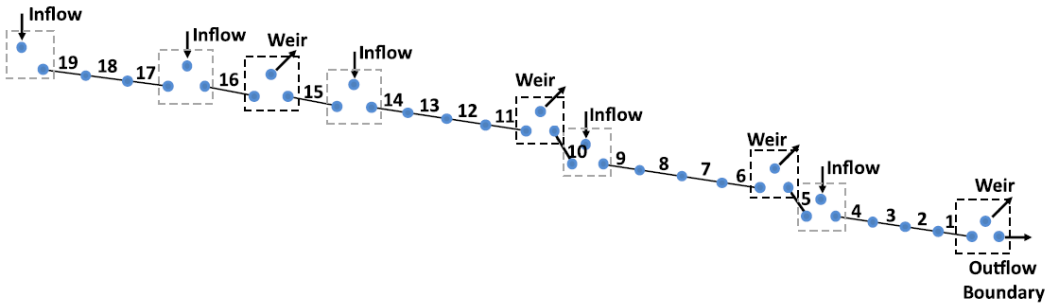


Figure 1.2: Example of the division of a system

$$\Delta F = \left(\frac{\partial f_i}{\partial x_j} \right)_{ij}$$

The iteration of Newton-Raphson method is:

$$F = F_{prev} + (\Delta F)^{-1} + \Delta X$$

Until F and F_{prev} are close enough.

Chapter 2

SOME RESULTS WITH SRM

Before showing any work on accuracy and spacing criterion, it is interesting to see some results given by the SRM method.
The first two cases that will be shown were tested by Matthey Hoy and are presented in his thesis (Matthew A. Hoy, 2005) [**?**]. The third case will be a simplified version a canal with a more complex cross section and the last cases will be theoretical cases of pipes.

2.1 Wallingford Hydraulic Research Station

In the case of Wallingford Hydraulic Research Station (Matthew A. Hoy, 2005) [**?**], the channel is a circular 1ft-diameter pipe. The inflow hydrograph is triangular and the outfall is a free overfall, here are the properties of the simulation:

Wallingford Hydraulic Research Station		Boundary Conditions	
Channel Properties		upstream: triangular inflow hydrograph	
geometry	circular	duration	132 s.
diameter	1 ft.	base flow	0.176 cfs.
slope	0.001	peak time	60 s.
Manning's roughness coefficient	0.0116	peak flow	0.66 cfs.
length	1,136 ft.	peak duration	12 s.
		downstream: free overfall	

Similarly to Hoy's work, for routing in the SRM, a time step of 2.5 s. and a spacing of 28.4 ft. are taken. The results are compared his results obtained

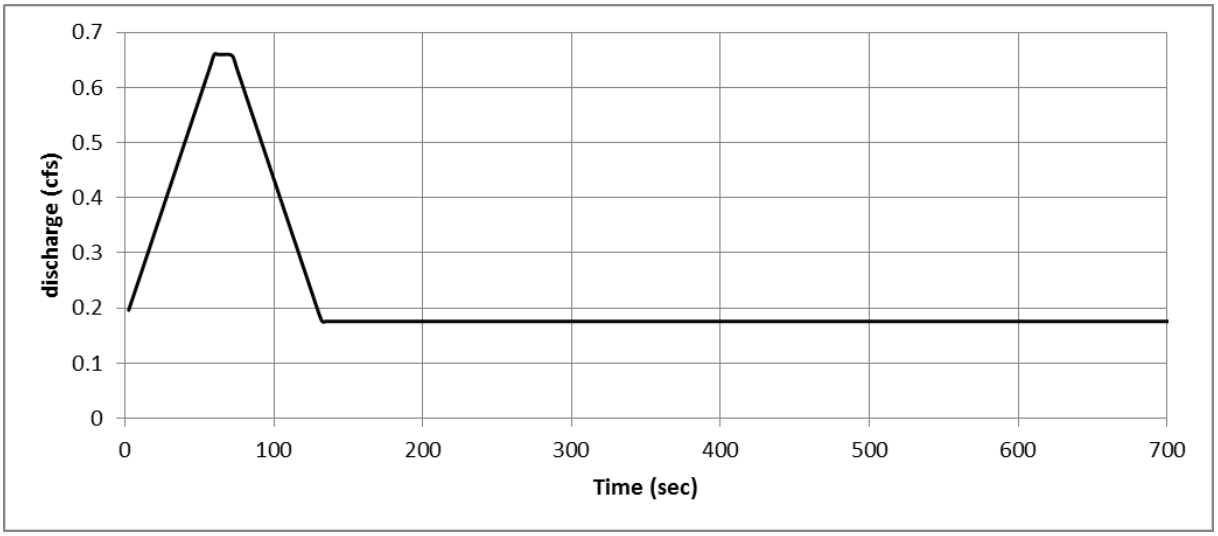


Figure 2.1: Inflow hydrograph for Wallingford Hydraulic Research Station

with FEQ and SRM and with the data from the report by Ackers and Harrison (1964).
The results for the water surface elevation at the stations 28.4, 255.7, 483, and 710 ft downstream of the entrance of the pipe are compared.

The plots show that the results found with SRM are the same as the results found previously (Matthew A. Hoy, 2005) [**?**]. And they match closely the experimental results.

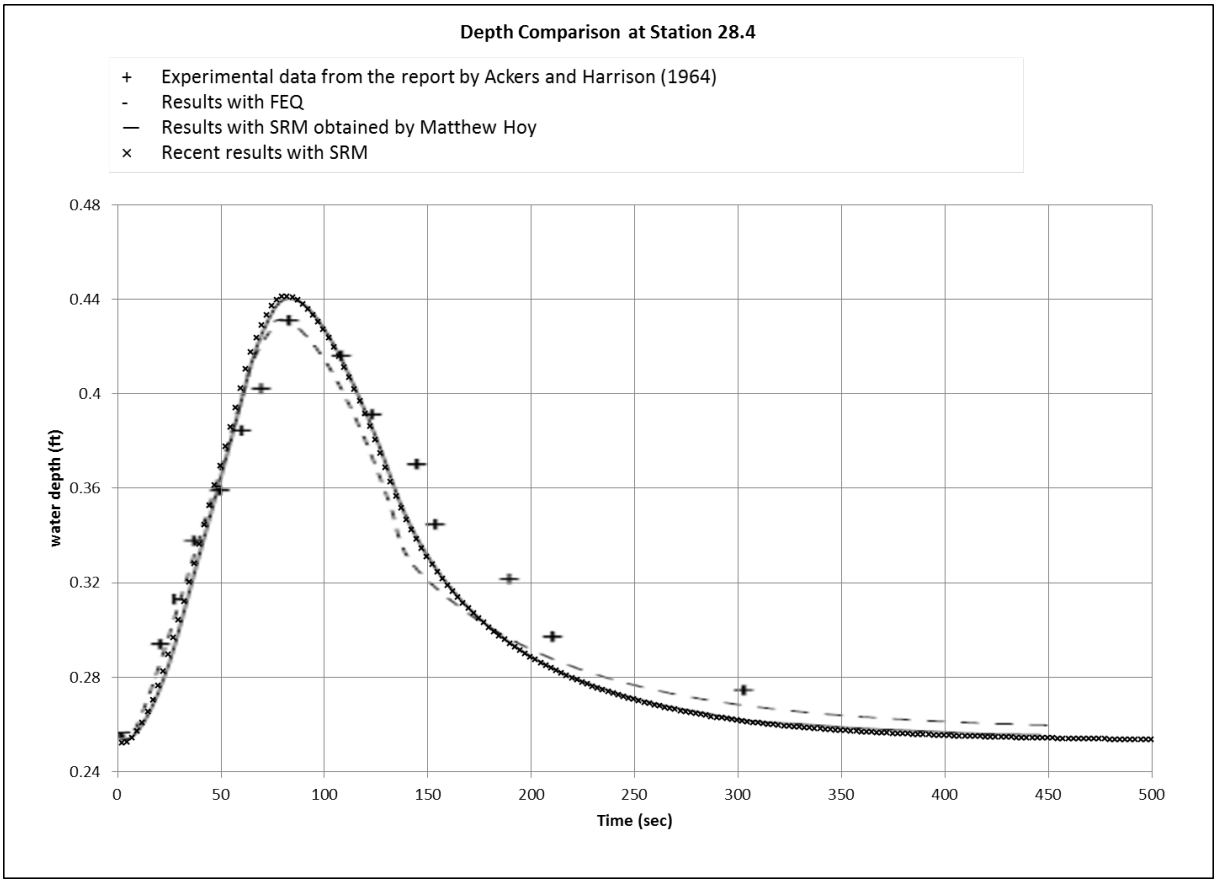


Figure 2.2: Water surface elevation at 28.4 ft. downstream of the entrance of the pipe

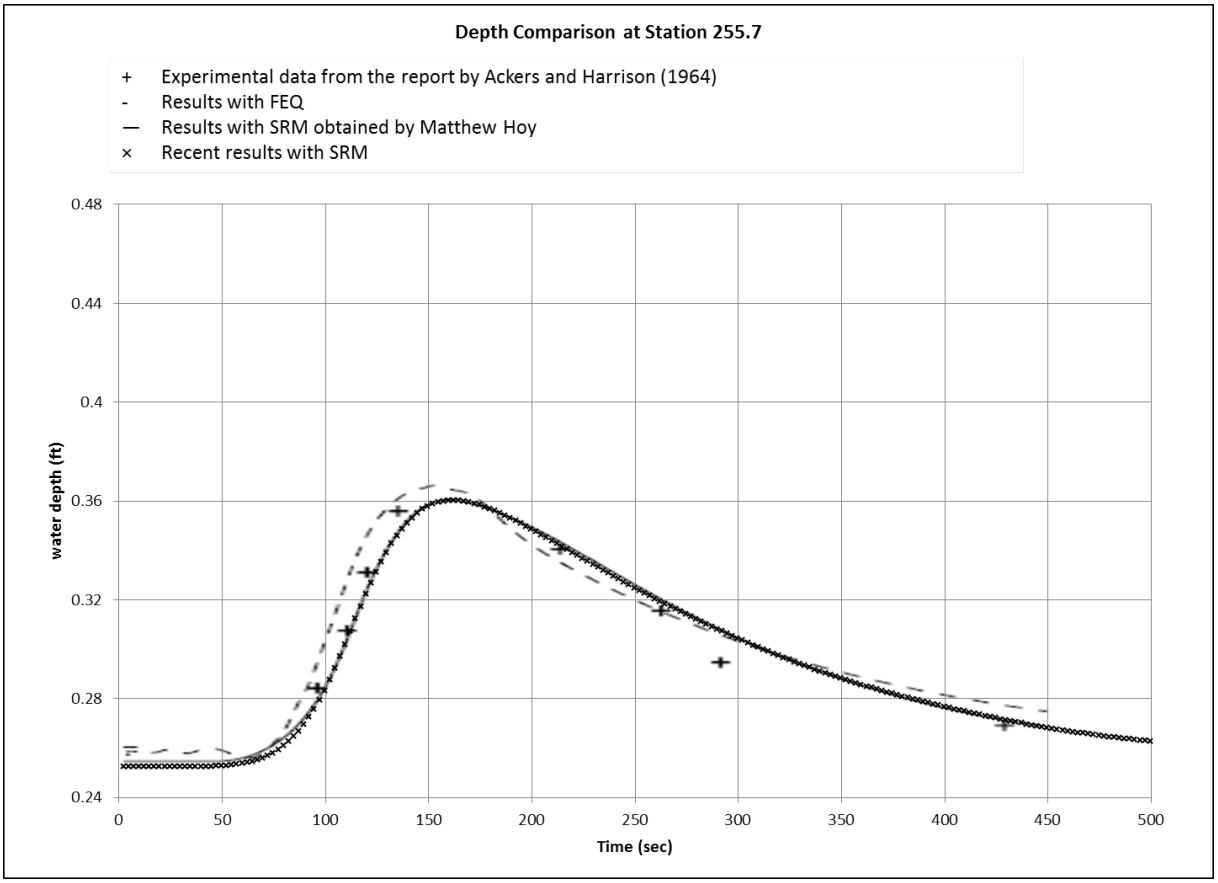


Figure 2.3: Water surface elevation at 255.7 ft. downstream of the entrance of the pipe

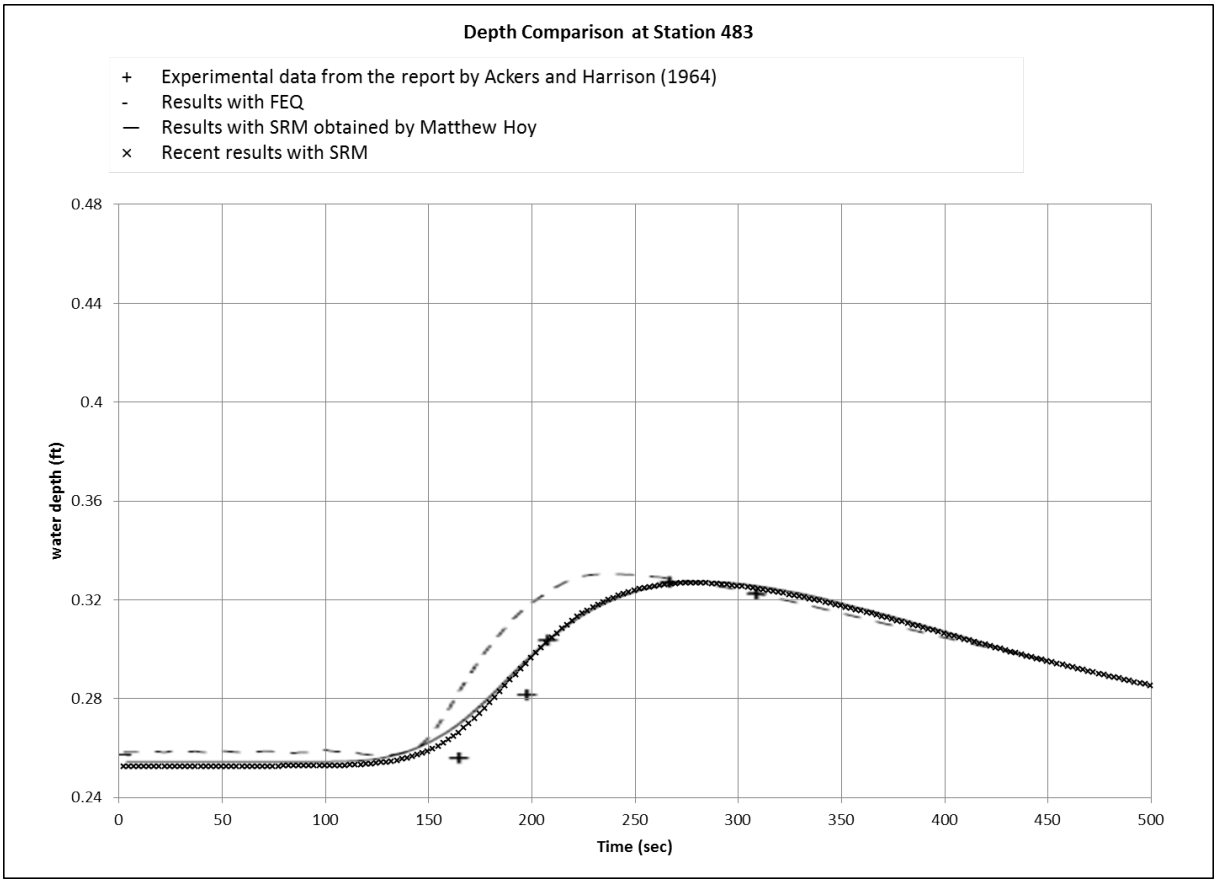


Figure 2.4: Water surface elevation at 483 ft. downstream of the entrance of the pipe

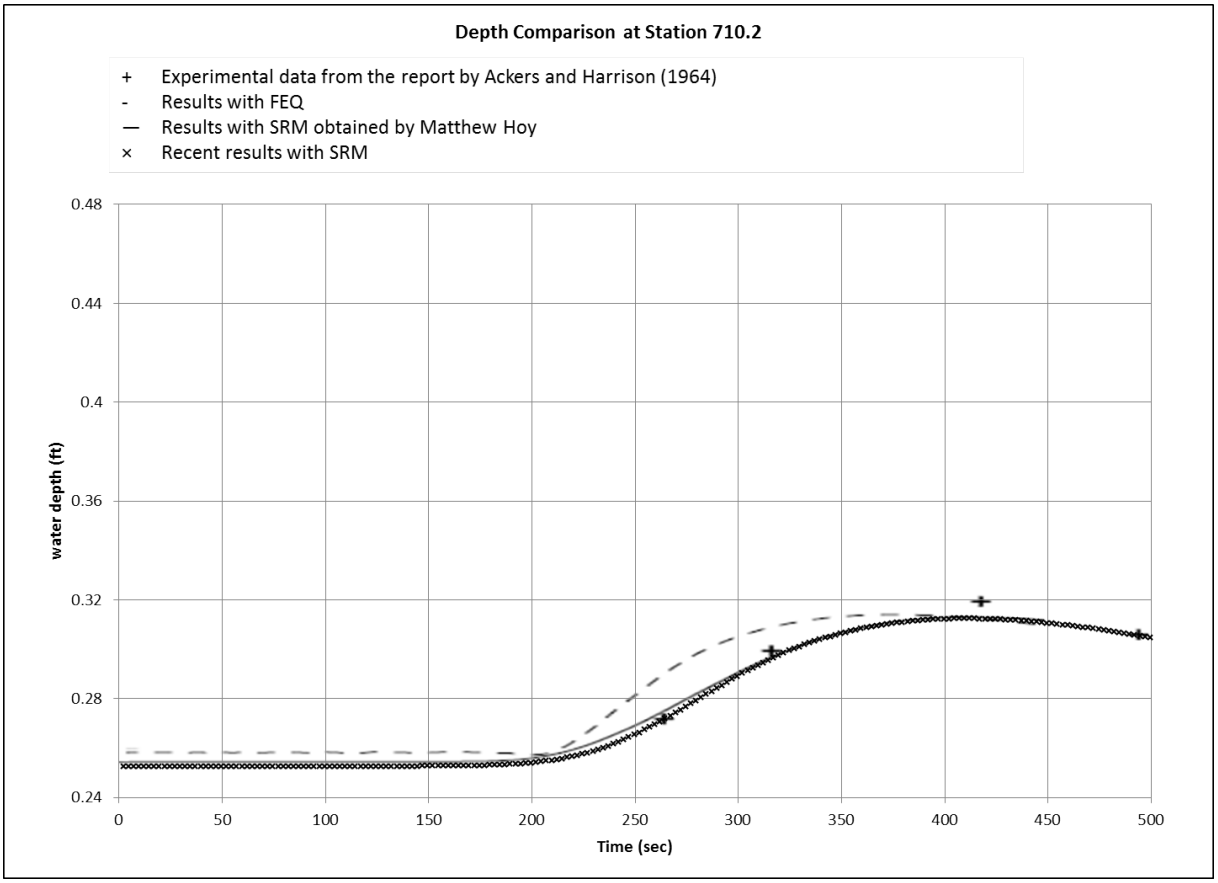


Figure 2.5: Water surface elevation at 710.2 ft. downstream of the entrance of the pipe

2.2 Theoretical Model from Matthew Hoy's Thesis

In this case (Matthew A. Hoy, 2005) [**?**], the channel is a circular 5ft-diameter pipe. The inflow hydrograph is triangular and the outfall is a free overfall, here are the properties of the simulation:

Theoretical Model from Matthew Hoy's Thesis		Boundary Conditions	
Channel Properties		upstream: triangular inflow hydrograph	
geometry	circular	duration	20 min.
diameter	5 ft.	base flow	10 cfs.
slope	0.0005	peak time	10 min.
Manning's roughness coefficient	0.015	peak flow	60 cfs.
length	5,000 ft.	peak duration	0 min.
		downstream: free overfall	

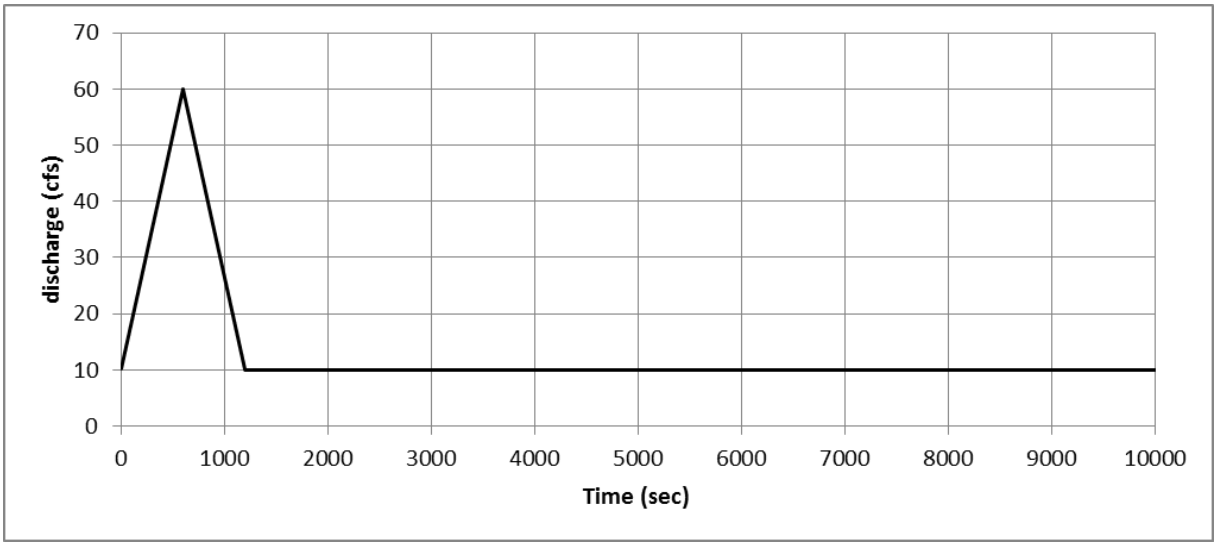


Figure 2.6: Inflow hydrograph for Theoretical Model from Matthew Hoy's Thesis

For routing in the SRM, a time step of 3 s. and a spacing of 100 ft. are taken. The results are compared his results obtained with FEQ and SRM. The results for the water surface elevation and the discharge at the stations 0, 2500, and 5000 ft downstream of the entrance of the pipe are compared.

In this case also, the plots show that the results found with SRM are the same as the results found previously.

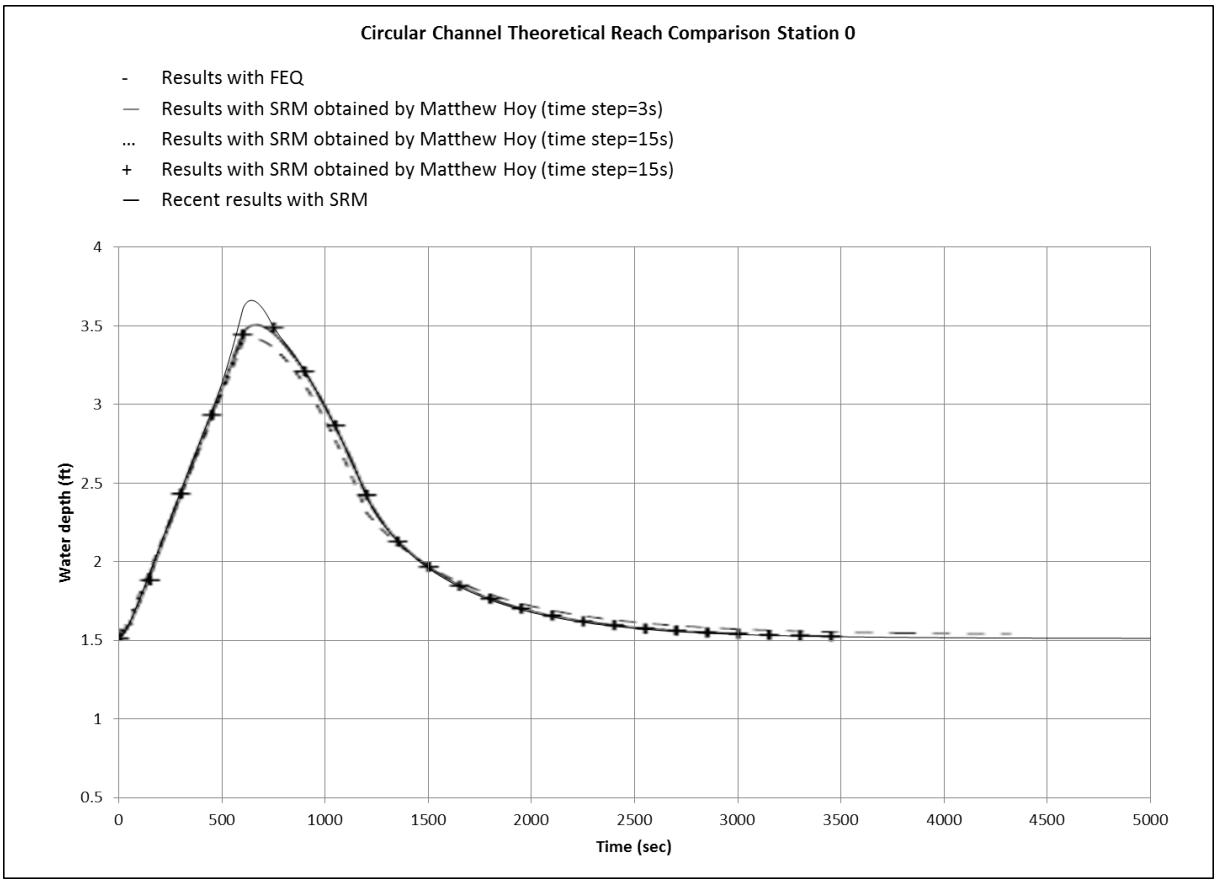


Figure 2.7: Water surface elevation at the entrance of the pipe

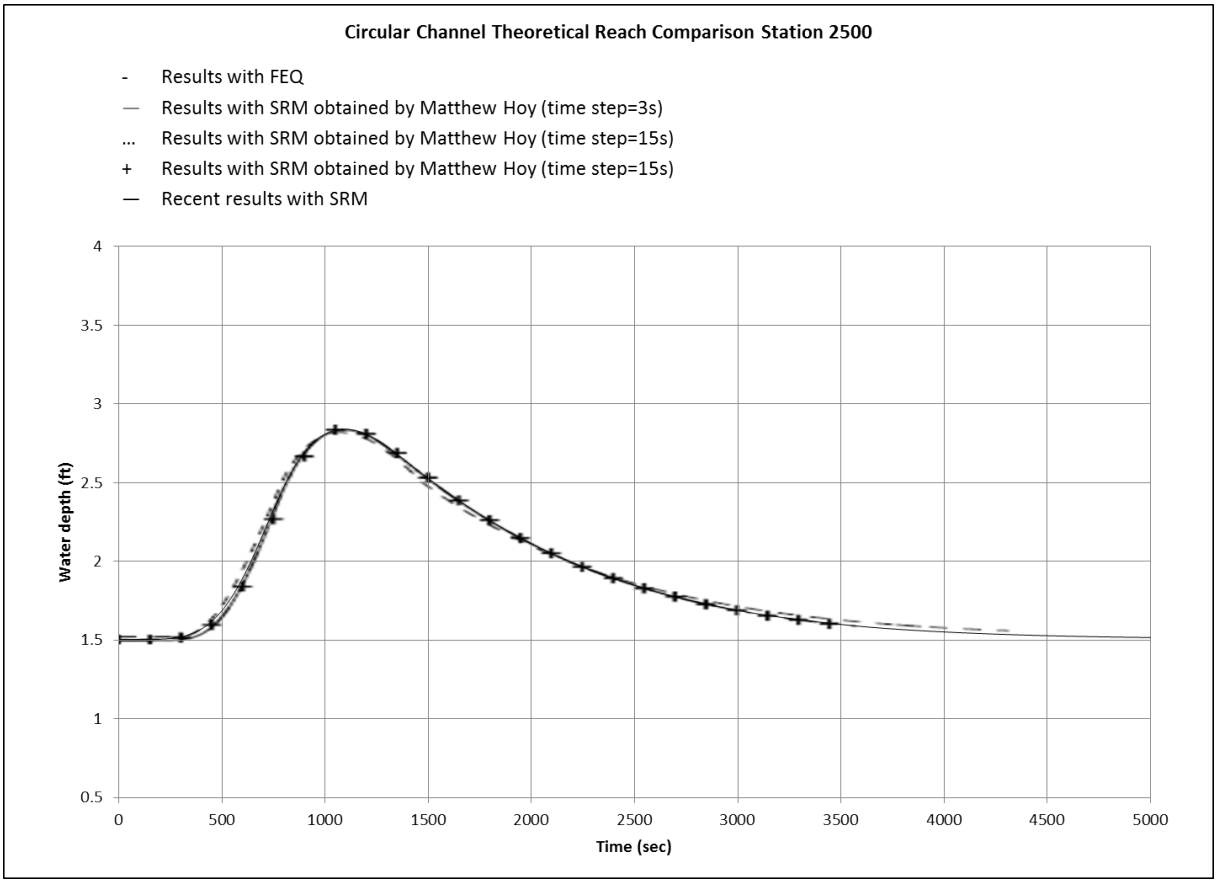


Figure 2.8: Water surface elevation at 2500 ft. downstream of the entrance of the pipe

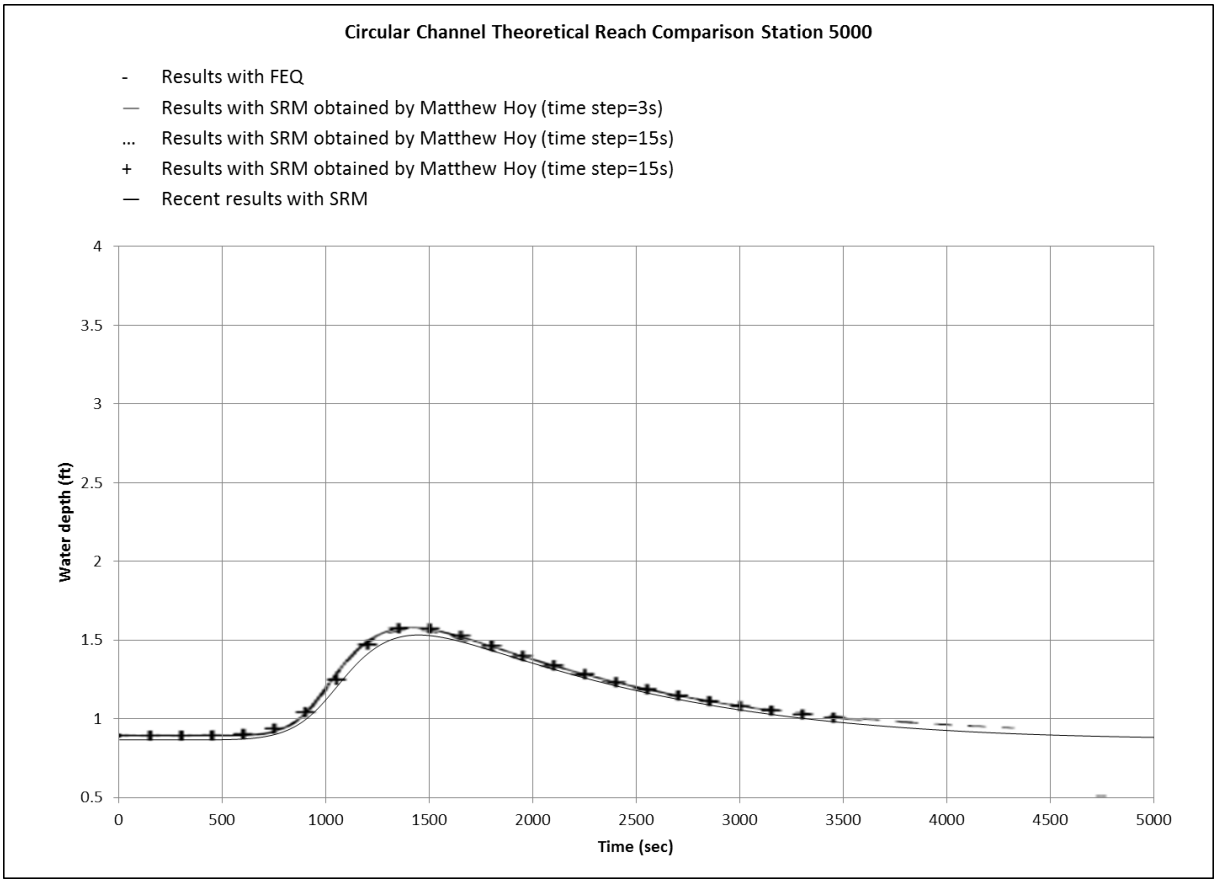


Figure 2.9: Water surface elevation at 5000 ft. downstream of the entrance of the pipe

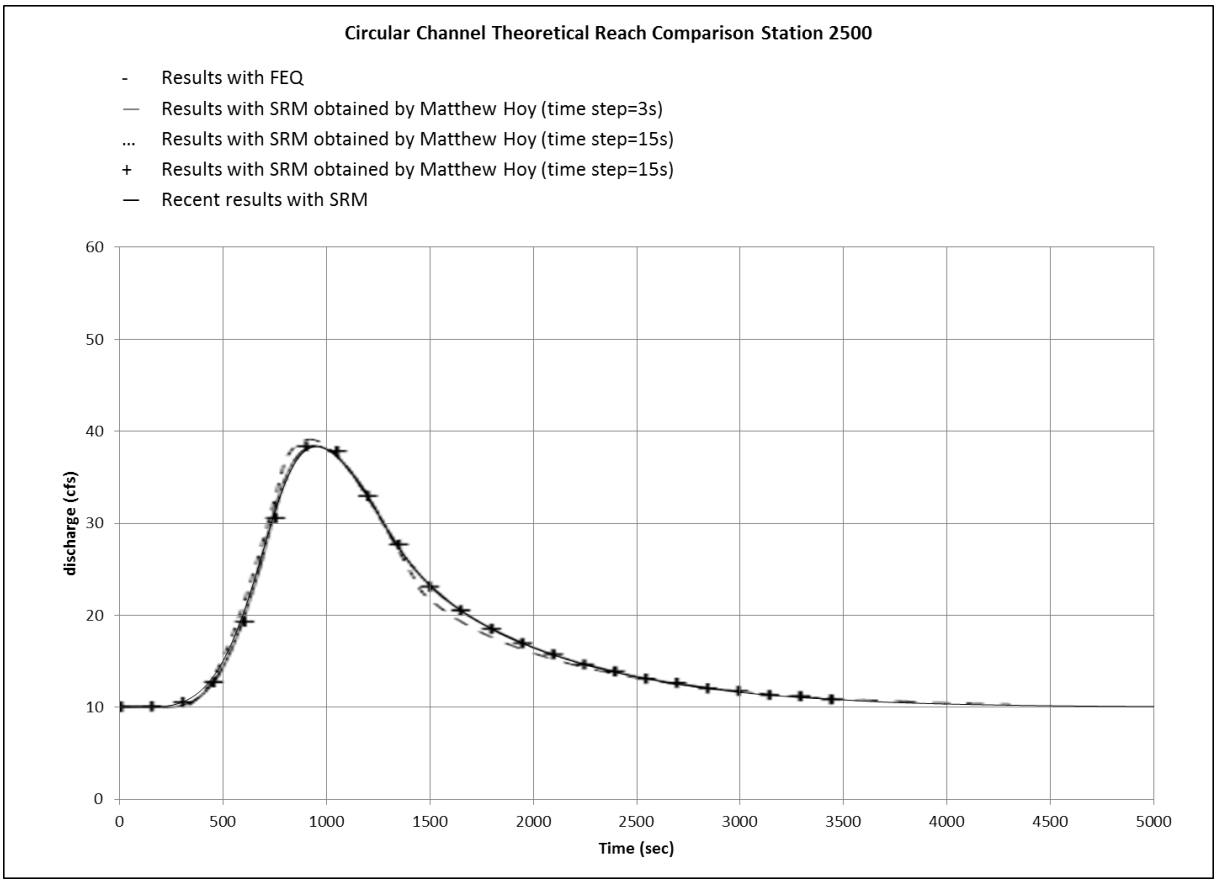


Figure 2.10: Discharge at 2500 ft. downstream of the entrance of the pipe

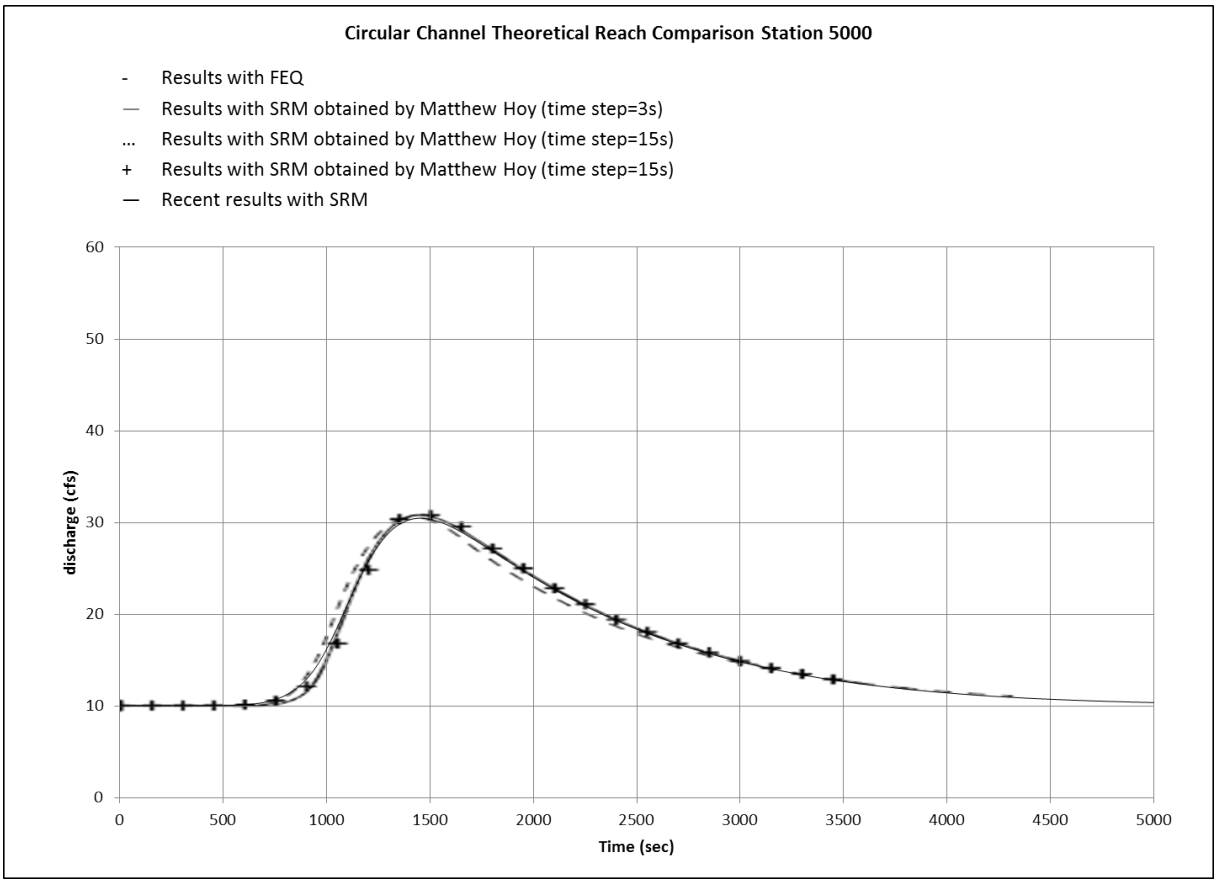


Figure 2.11: Discharge at 5000 ft. downstream of the entrance of the pipe

2.3 The Canal

In this case, the channel is prismatic with a more complex geometry, see 2.13. The inflow hydrograph is Gaussian distribution and the outfall is a free overfall, here are the properties of the simulation:

canal		Boundary Conditions	
Channel Properties		upstream: Gaussian distribution hydrograph	
geometry	prismatic	duration	1 hr. approx.
depth	25 m	base flow	7000 m^3/s
slope	0	peak time	1 hr.
Manning's roughness coefficient	0.0286	peak flow	14000 m^3/s
length	16,000 m	downstream: free overfall	

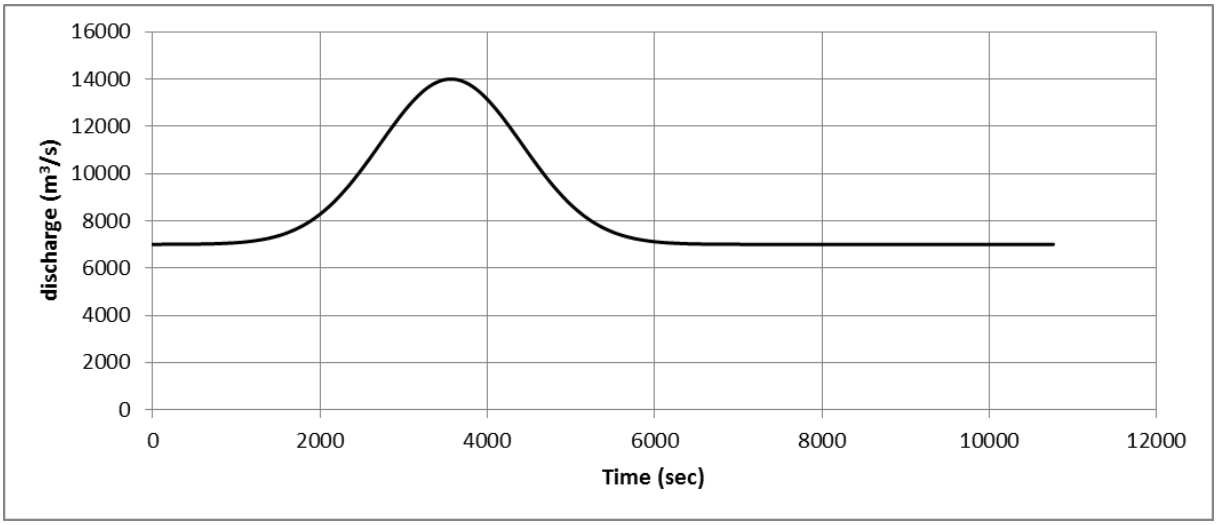


Figure 2.12: Inflow hydrograph for The Canal

For routing in the SRM, a time step of 36 s. and a spacing of 160 m are taken. The simulation is also done with HEC-RAS.
The results for the water surface elevation and the discharge at the upstream and downstream end.

The results given by SRM and by HEC-RAS are very close in this case. The maximum error for this simulation is $\epsilon = 4.43\%$.

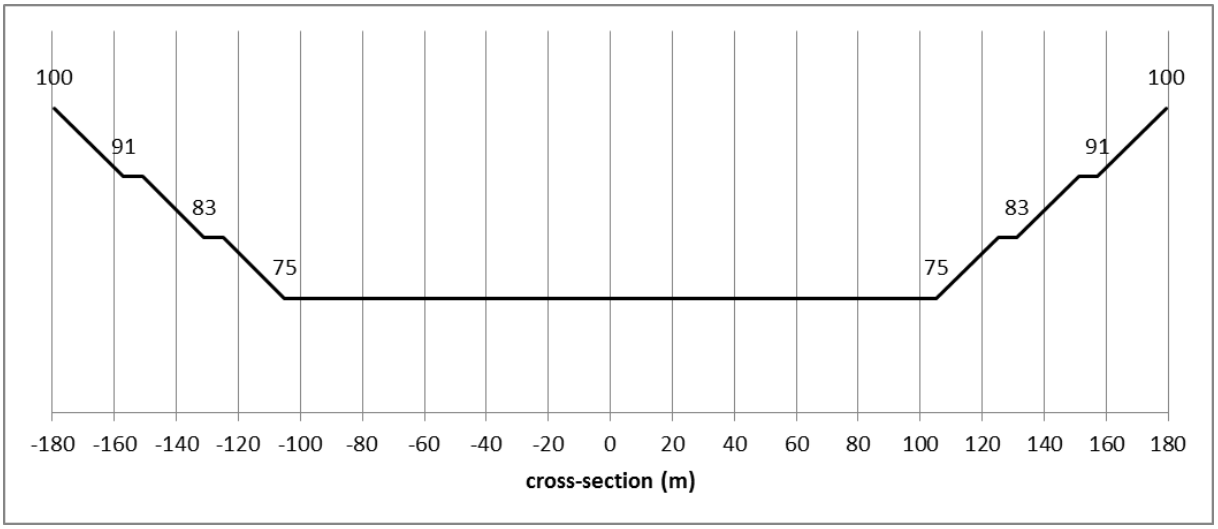


Figure 2.13: Cross-section geometry of The Canal

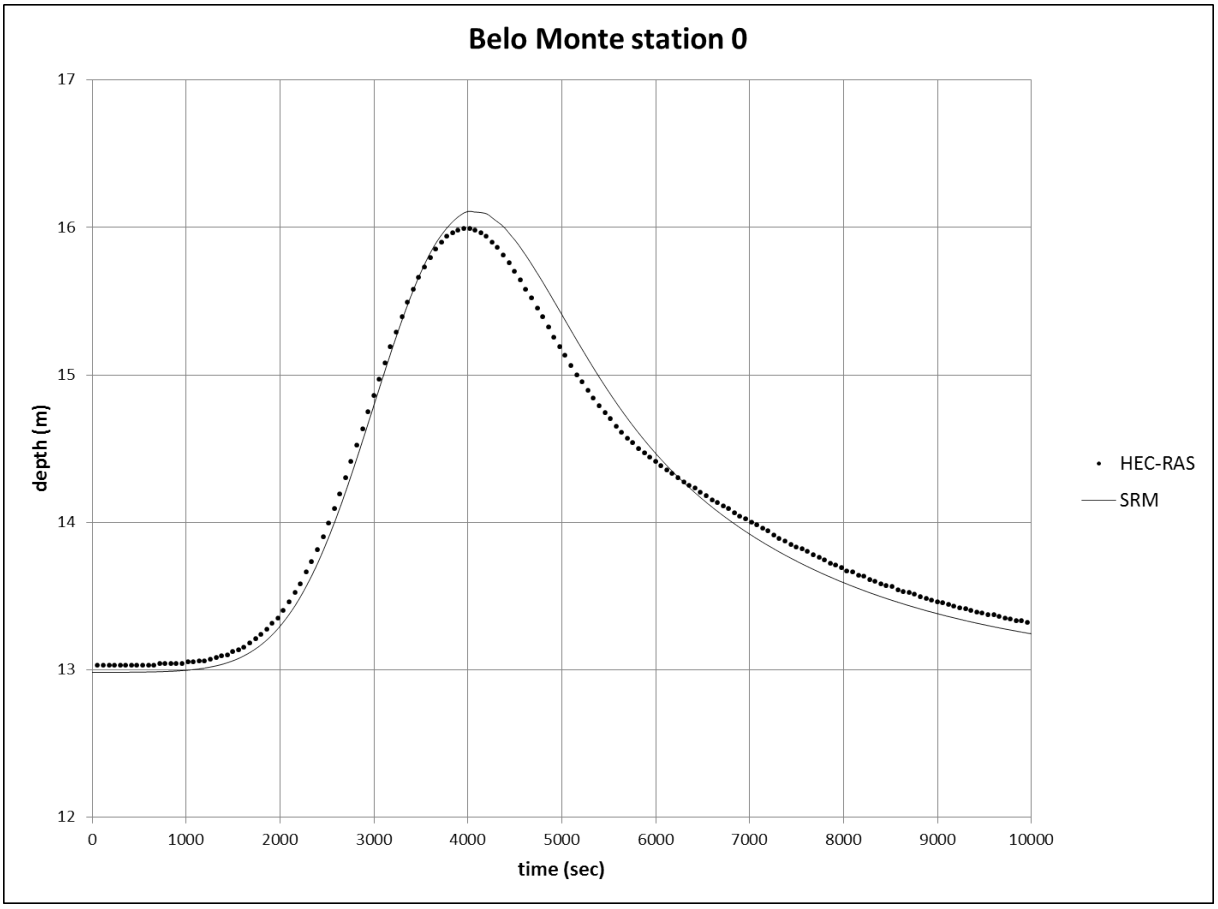


Figure 2.14: Water surface elevation at the entrance of the canal

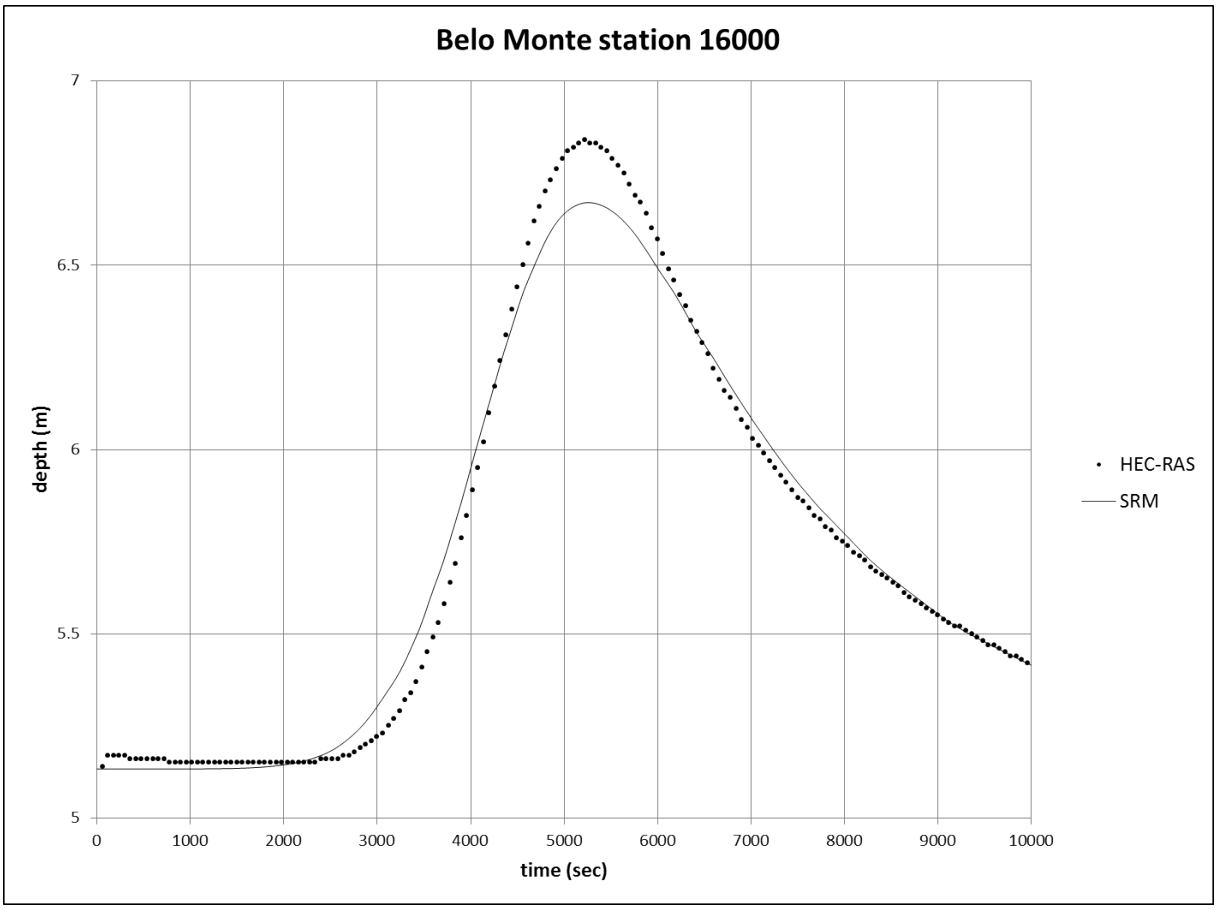


Figure 2.15: Water surface elevation at the exit of the canal

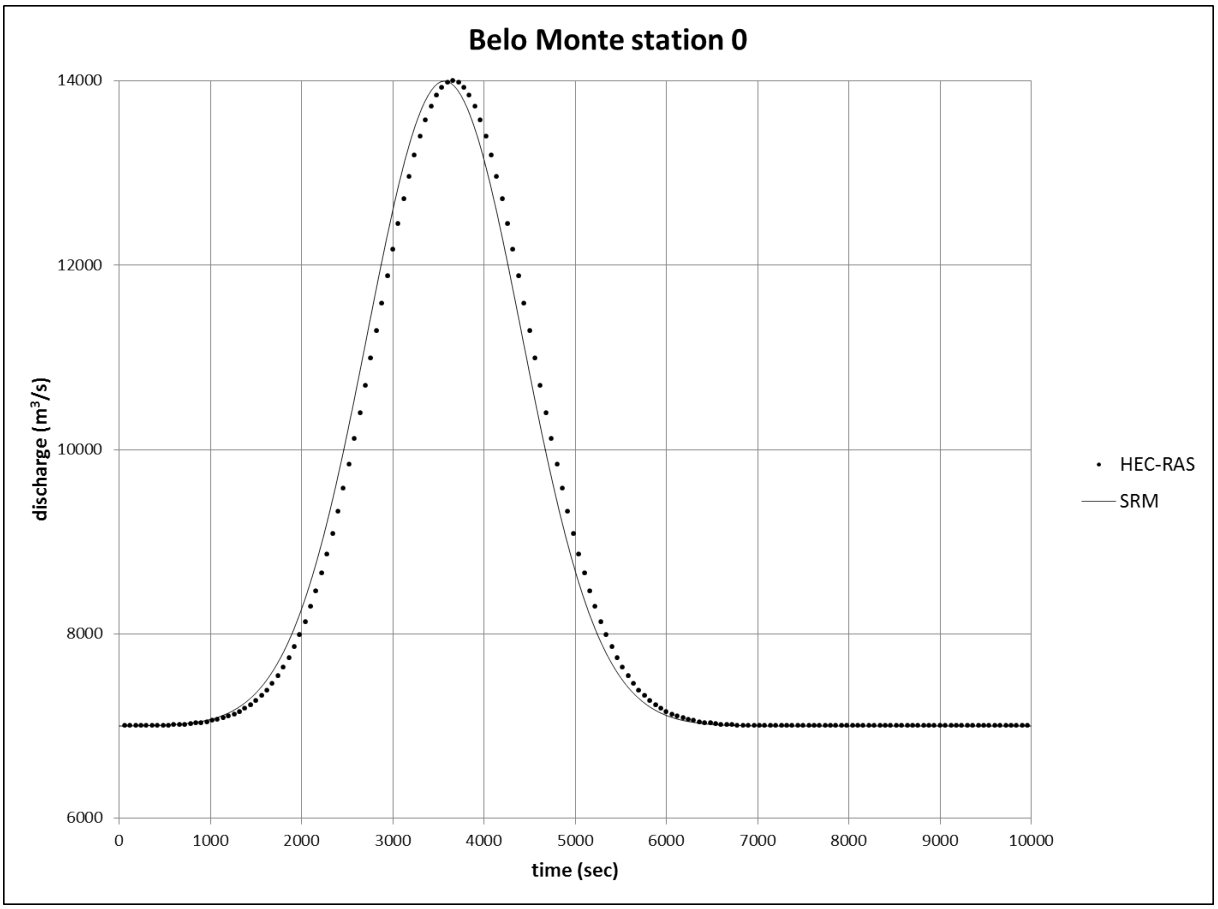


Figure 2.16: Discharge at the entrance of the canal

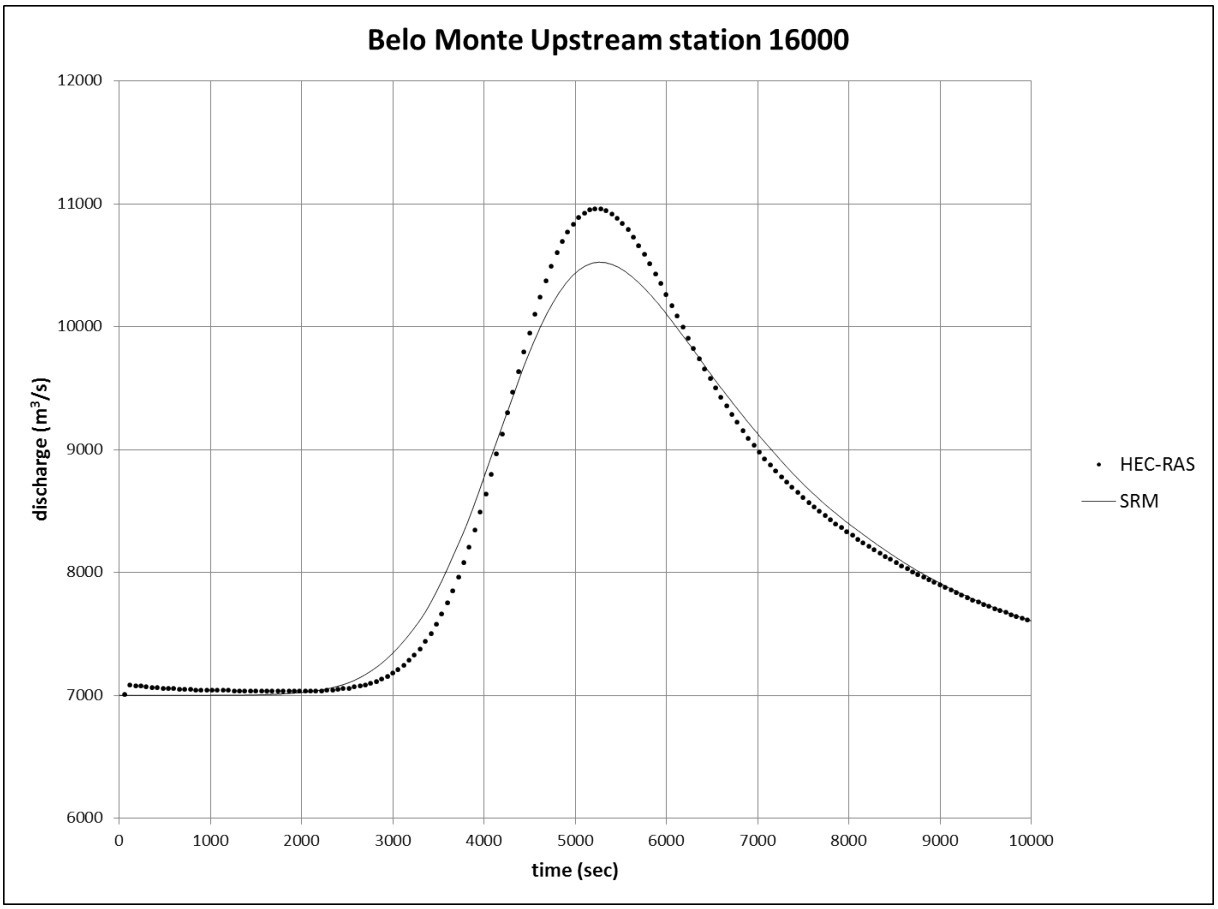


Figure 2.17: Discharge at the exit of the canal

2.4 The Chicago River Main Stem

In this case, the calculation is done for the Main Stem part of the Chicago River as seen in the picture 2.18.

Figure 2.18: View of the Chicago River Main Stem

the geometry is not prismatic:

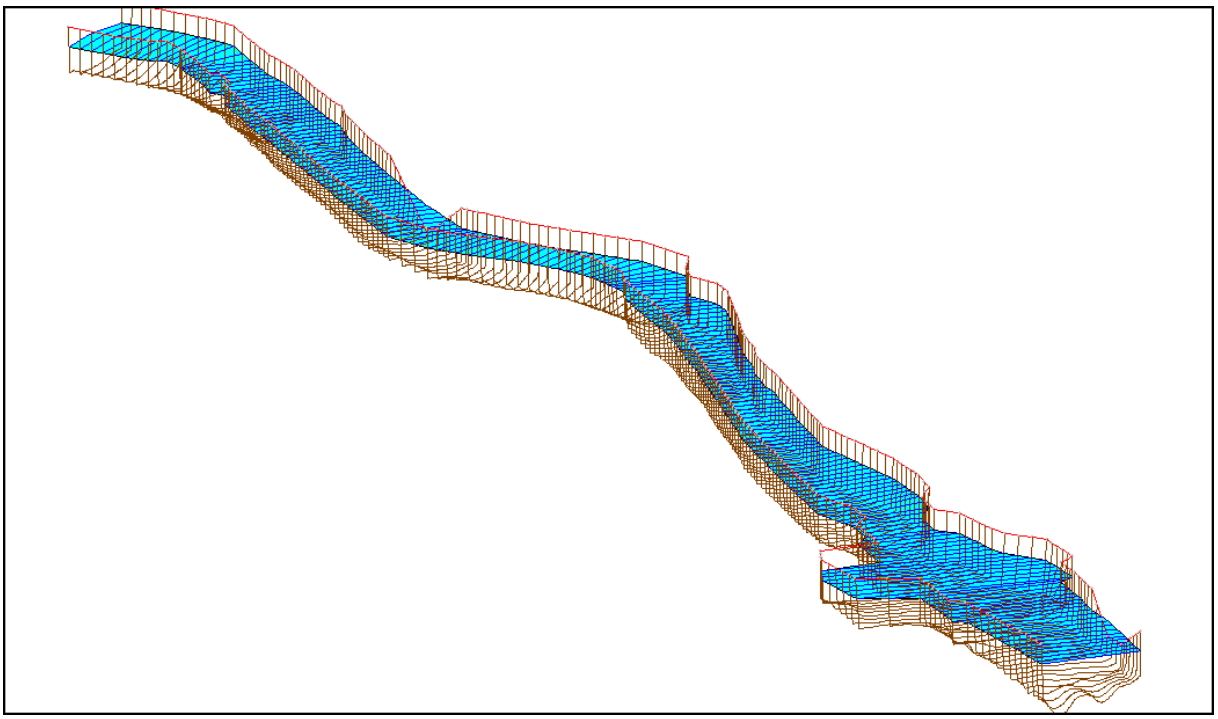


Figure 2.19: Geometry of the Chicago River Main Stem

The boundary condition at the upstream end is a stage hydrograph and at the downstream end is a rating curve:

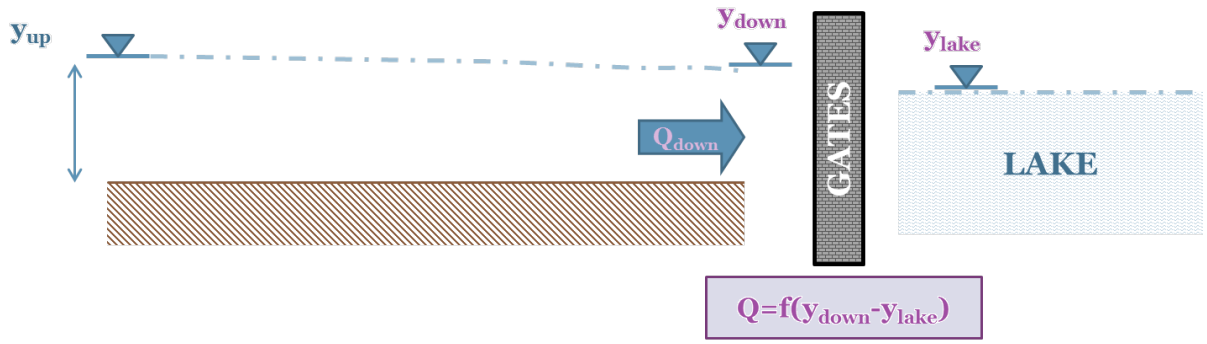


Figure 2.20: Geometry of the Chicago River Main Stem

Chicago river Main Stem		Boundary Conditions	
Channel Properties		upstream: stage hydrograph	
geometry	prismatic	duration	5 days approx.
depth	around 20 ft.	base flow	$y = 0ft.$
Manning's roughness coefficient	0.03	peak flow	$y = 3ft.$
length	1.3280 miles	downstream: rating curve	

2.4.1 Rating curve

The rating curve used in this case is the one for all gates opened:

$$Q = 18694\sqrt{\Delta y}$$

2.4.2 Stage hydrograph

The data given by the USGS website (http://waterdata.usgs.gov/nwis) is a stage hydrograph for the upstream point. This hydrograph contains a lot of steep osculations.

When using this hydrograph as a boundary condition, the model crashes. The osculations are too steep. therefore it is necessary to use a smoother hydrograph:

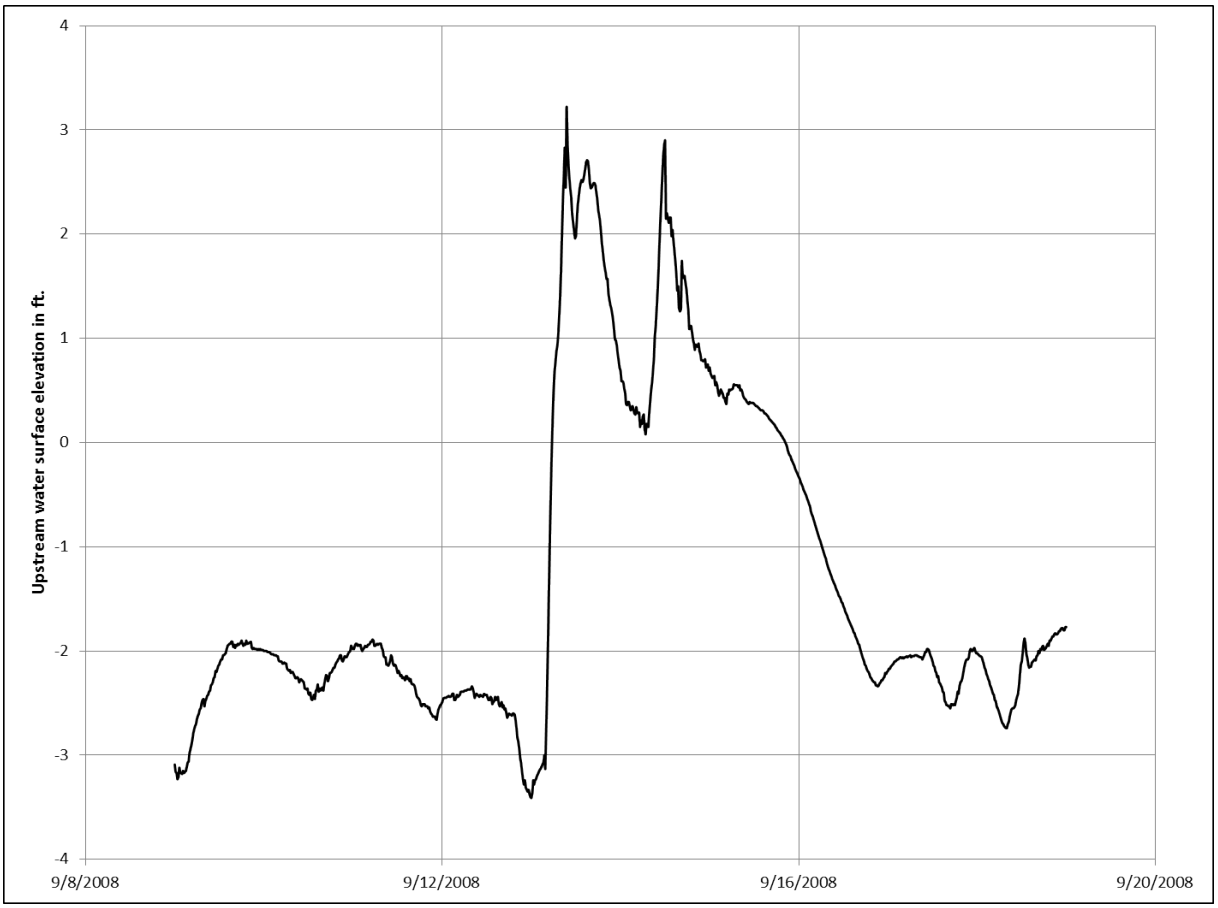


Figure 2.21: Stage hydrograph at the upstream point of Main Stem

2.4.3 Geometry

The channel cannot be modeled as a circular conduit. However, the code is written to create HPGs and VPGs for circular conduits. Thus, the HPGs and VPGs have to be created first and use as an input to the code.

2.4.4 results

The same simulation is done HEC-RAS to compare the results.

The error is calculated with the following formula:

$$\epsilon = \frac{\overline{|Q_{SRM} - Q_{HEC-RAS}|}}{Q_{Q_{SRM}+Q_{HEC-RAS}}}$$

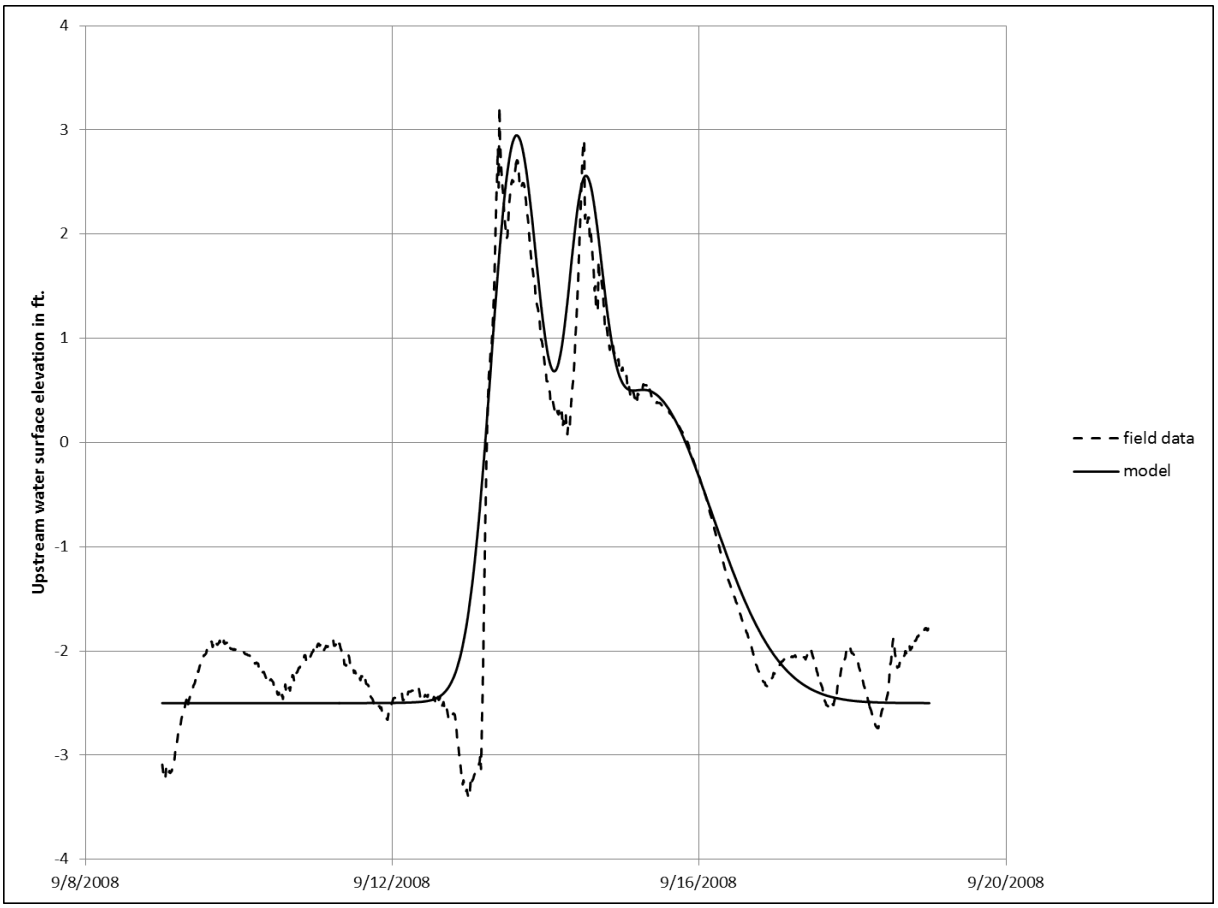


Figure 2.22: Model for the stage hydrograph at the upstream point of Main Stem

The results given by SRM and by HEC-RAS are very close in this case. The error always stays below 10% except for the first steps. The error at the first steps might be caused by the initial condition in HEC-RAS.

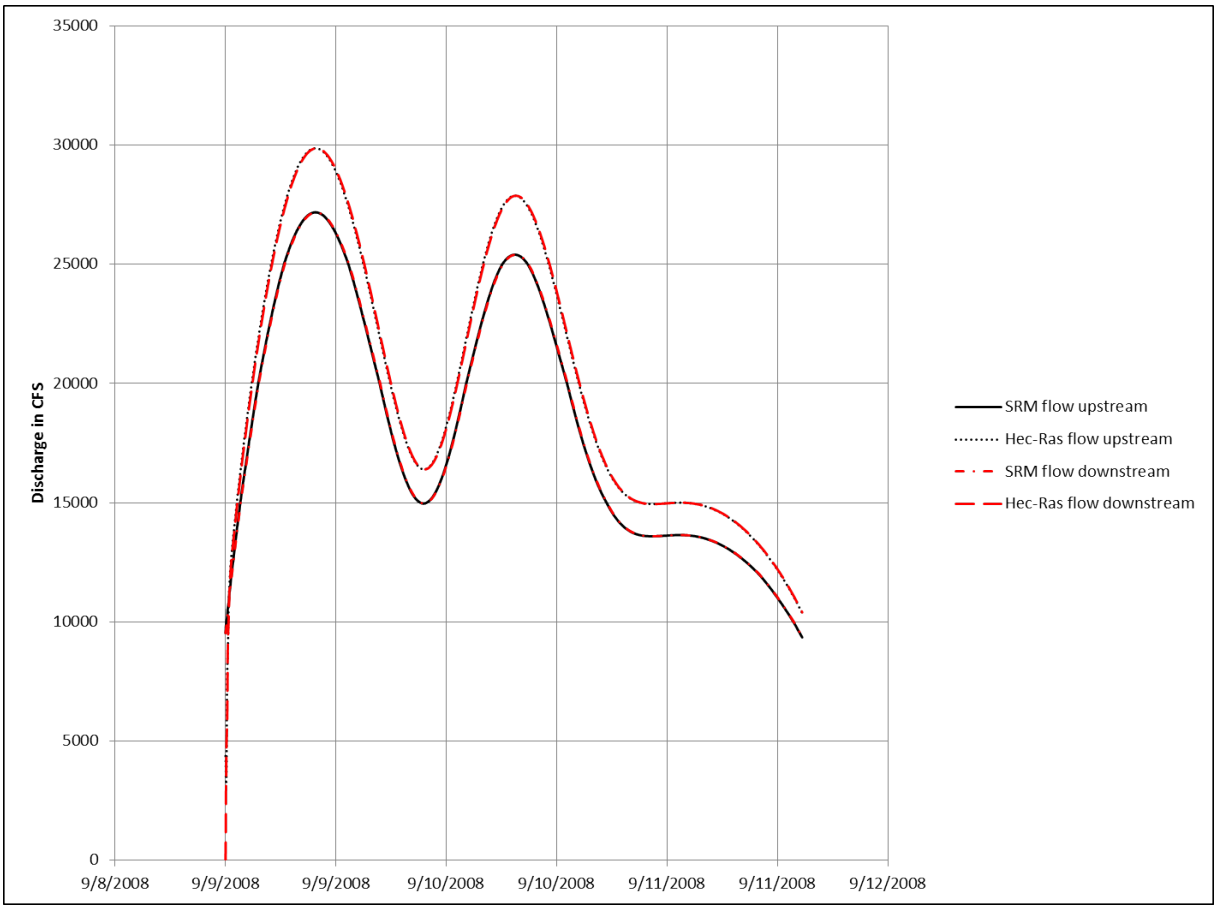


Figure 2.23: Discharge in the Main Stem

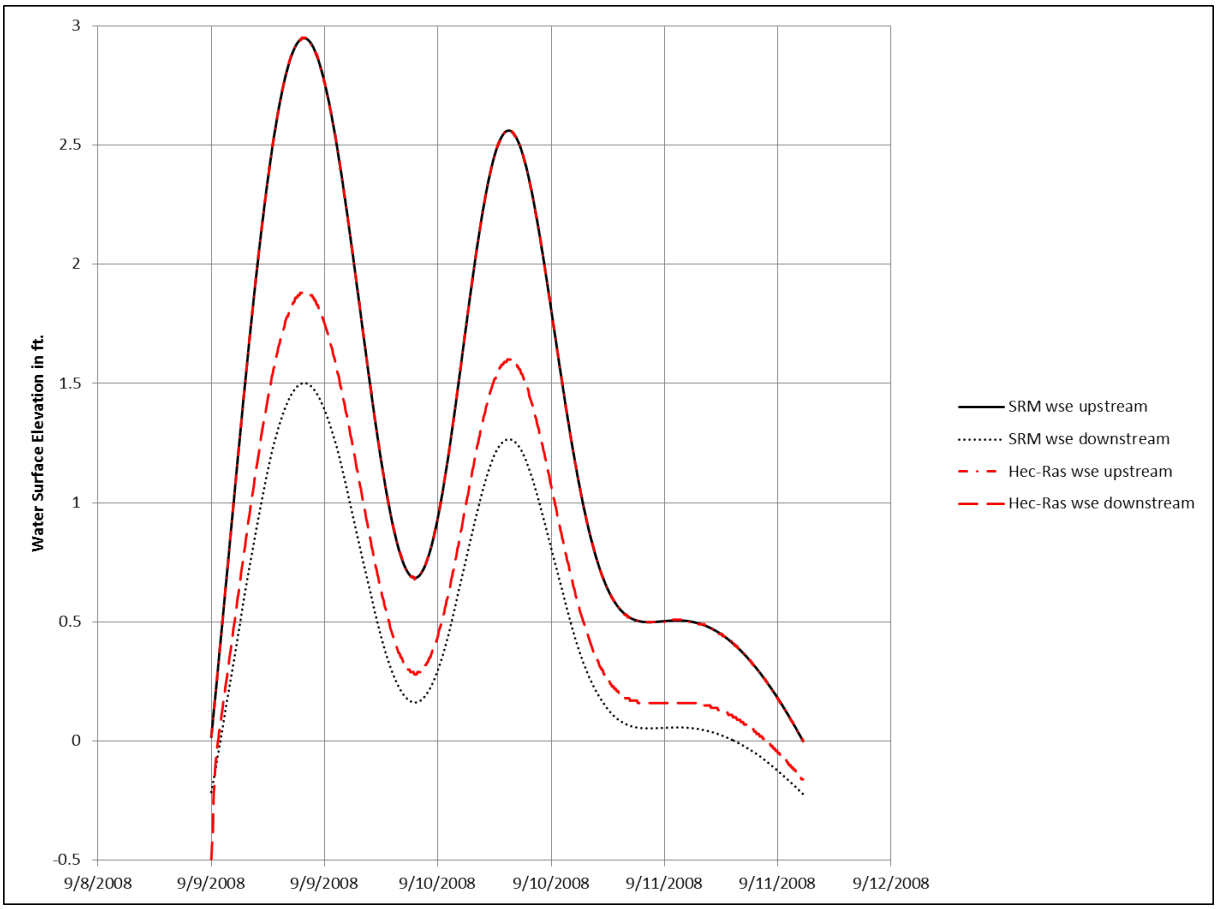


Figure 2.24: Water surface elevation in the Main Stem

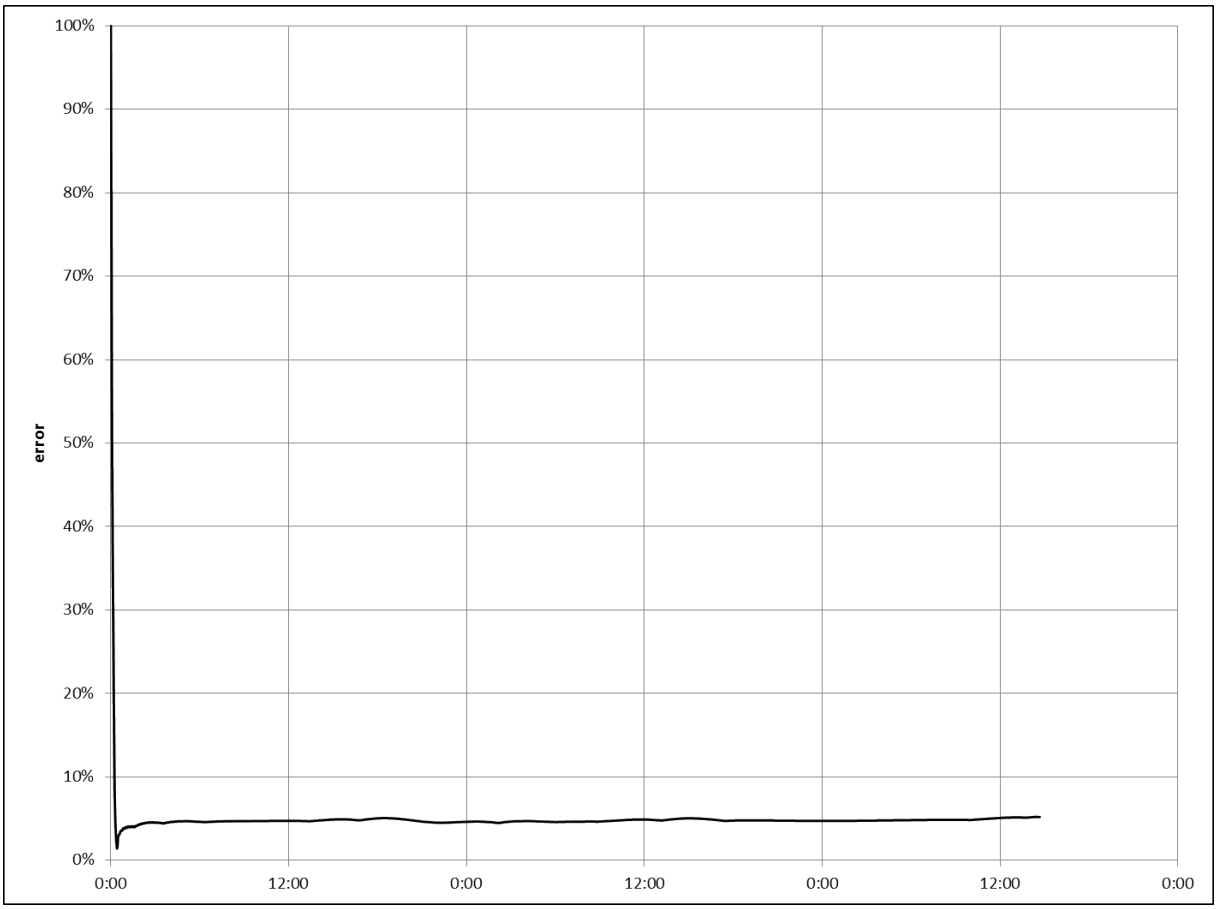


Figure 2.25: error between SRM and HEC-RAS results

2.5 Theoretical cases

In this case, the channel is a circular 50 ft-diameter pipe. Four different cases are tested, with two different slopes and two different inflow hydrographs. The inflow hydrograph has a gaussian distribution and the outfall is a free overfall, here are the properties of the simulation:

Theoretical cases		Boundary Conditions	
Channel Properties		upstream: Gaussian distribution hydrograph	
geometry	circular	duration	11 min and 2hr.
diameter	50 ft.	base flow	7000 cfs.
slopes	$2 \cdot 10^{-6}$,$2 \cdot 10^{-5}$	peak time	4 min and 40 min
Manning's n	0.01	peak flow	12000 cfs.
length	25,000 ft.	downstream: free overfall	

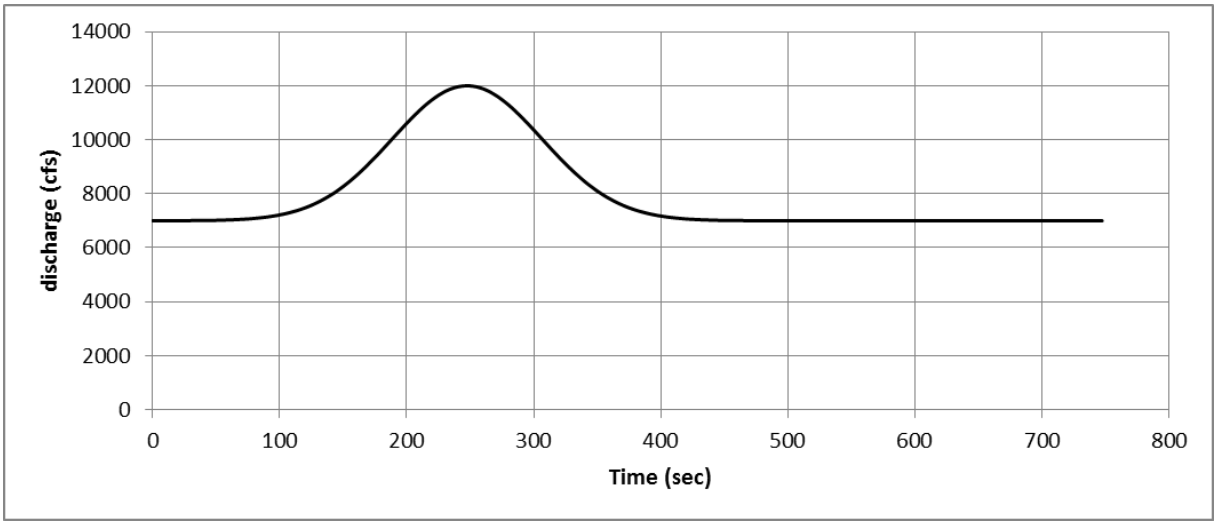


Figure 2.26: First Inflow hydrograph

For routing in the SRM, respective time step of 2.5 and 25 s. and a spacing of 250 ft. are taken.
The results for the water surface elevation and the discharge at the upstream and downstream end.

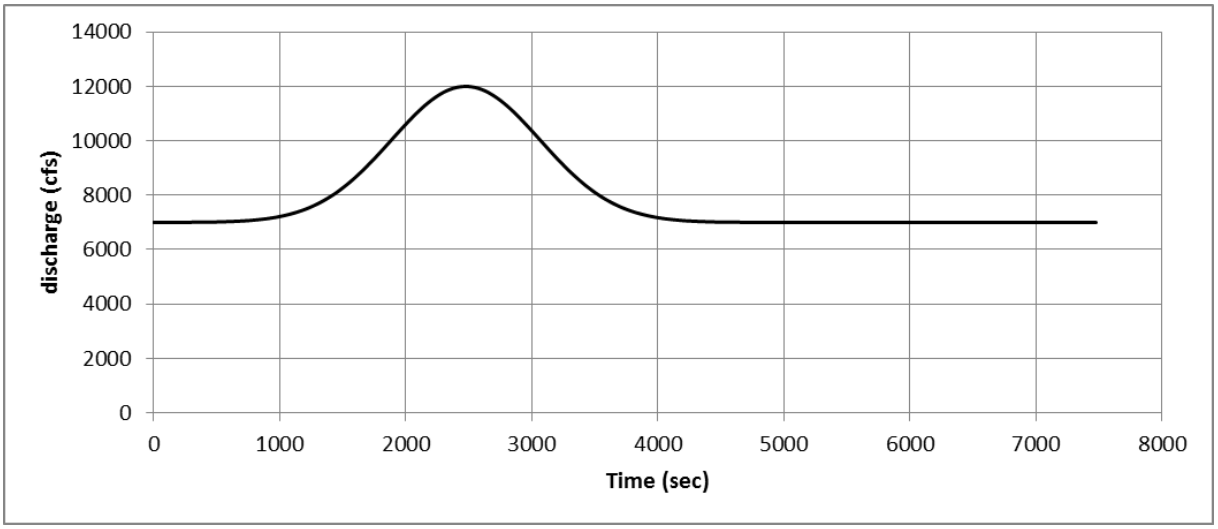


Figure 2.27: Second Inflow hydrograph

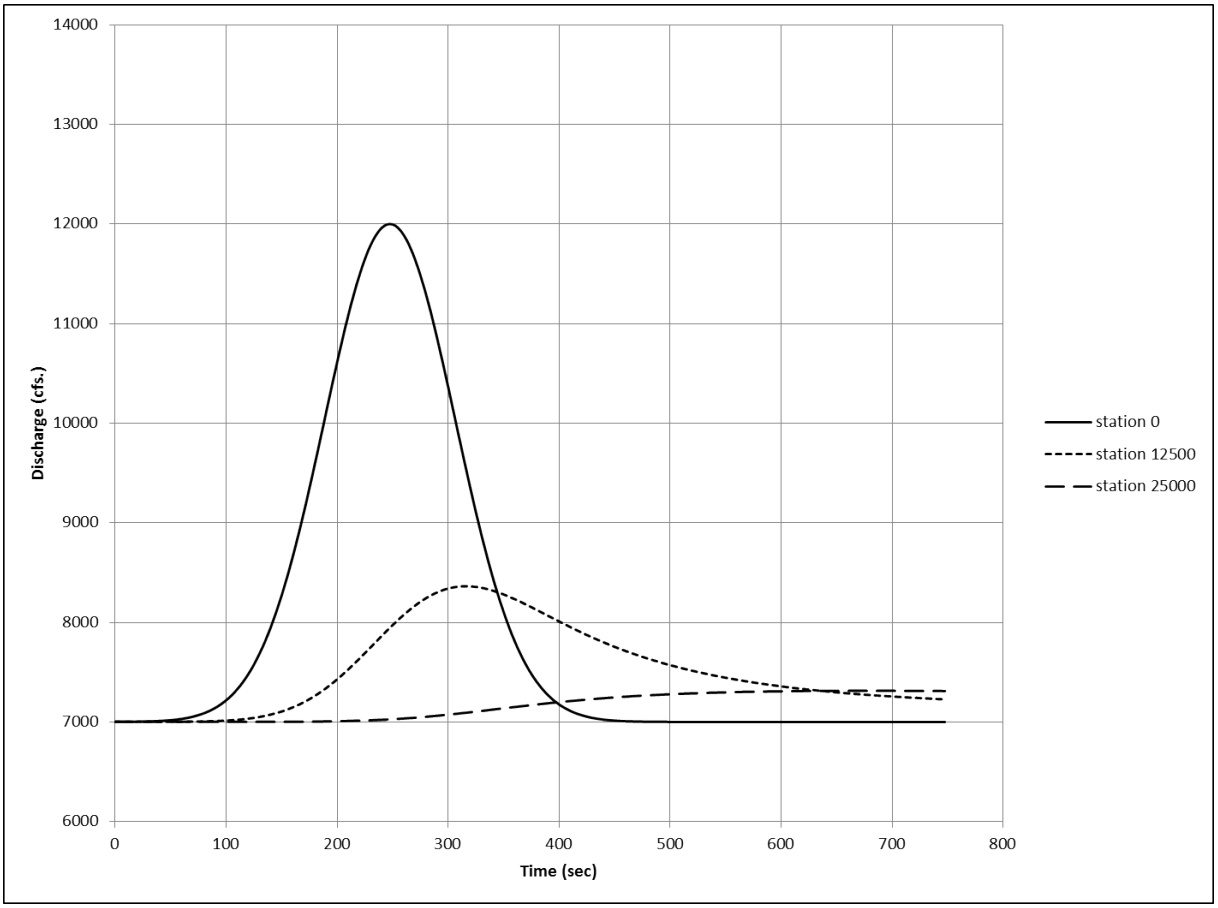


Figure 2.28: Discharge at several station for $S = 2 \times 10^{-6}$ with the first hydrograph

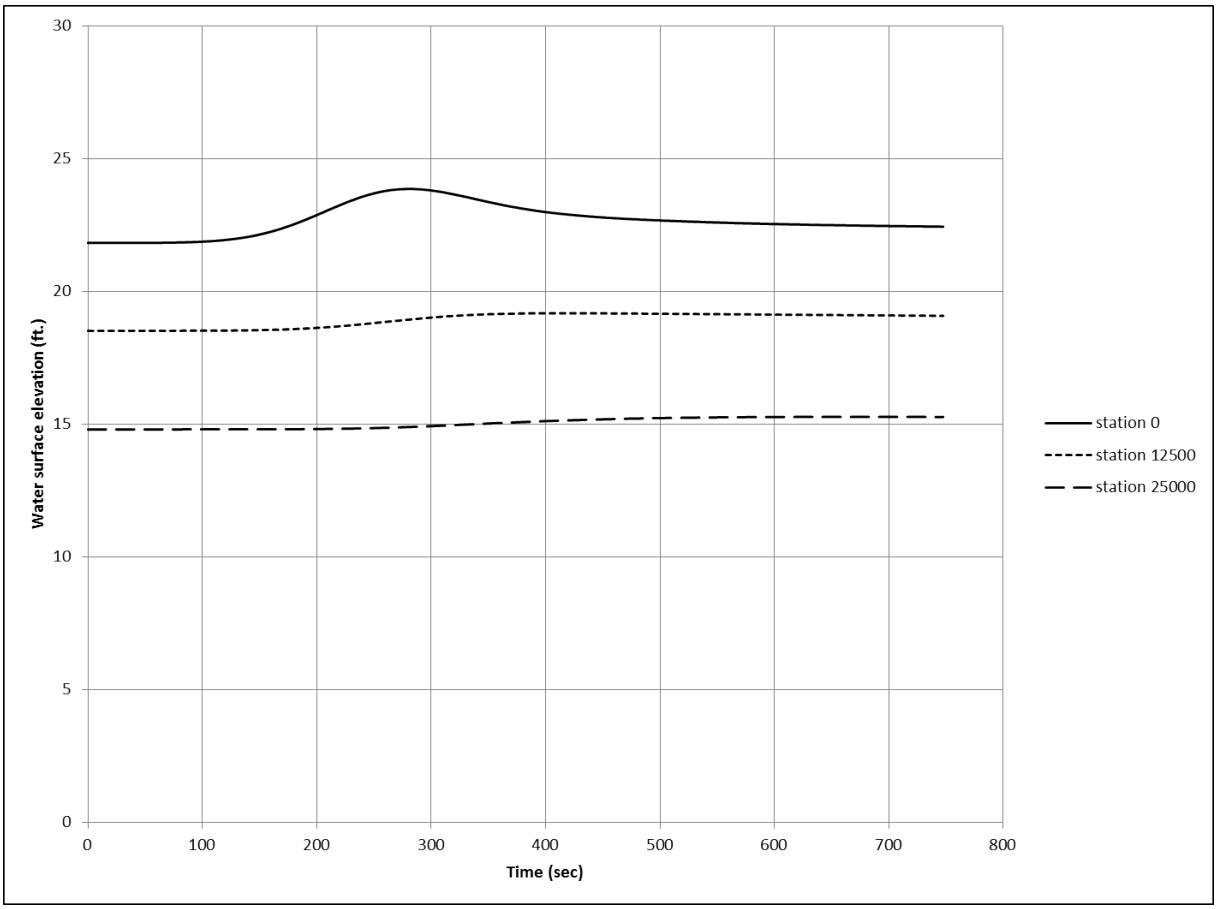


Figure 2.29: Water surface elevation at several station for $S = 2 \times 10^{-6}$ with the first hydrograph

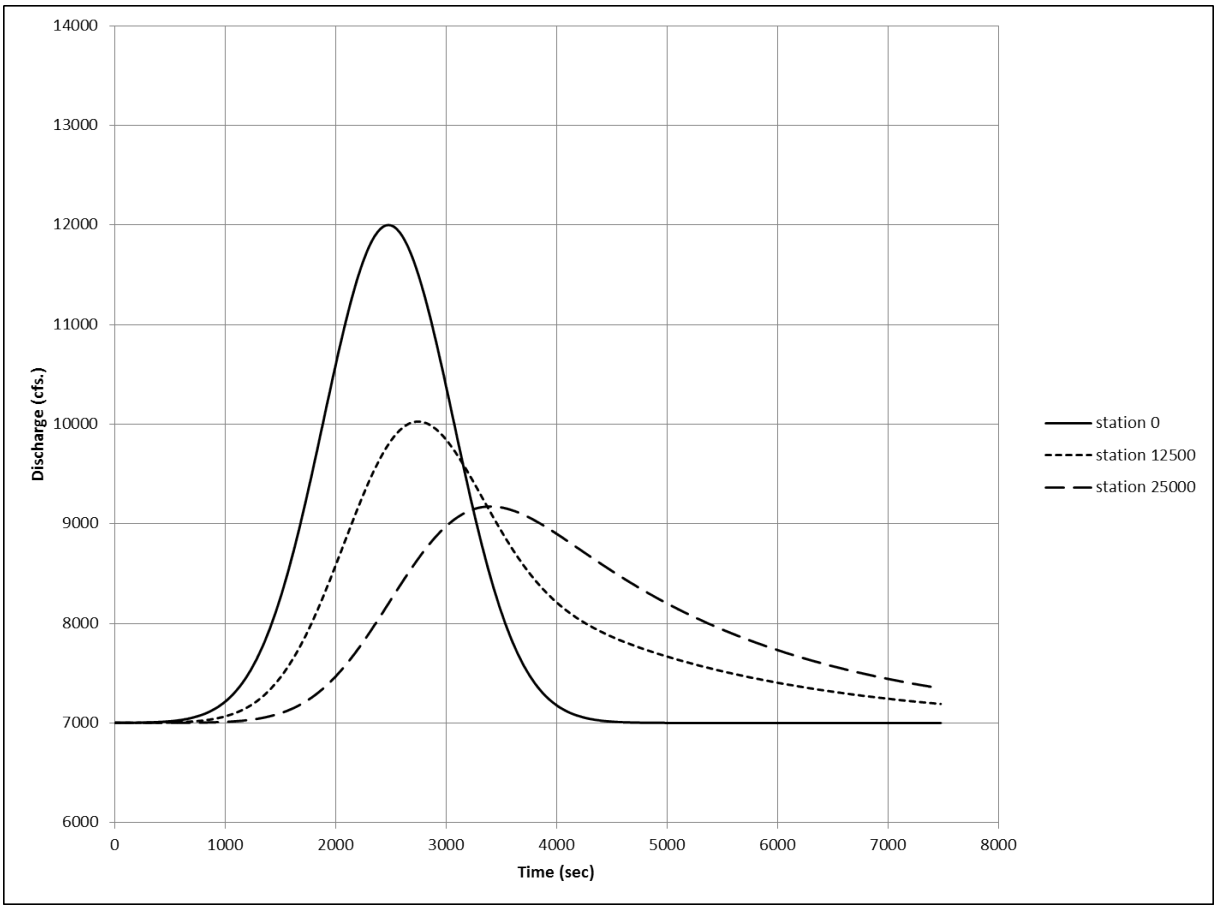


Figure 2.30: Discharge at several station for $S = 2 \times 10^{-6}$ with the second hydrograph

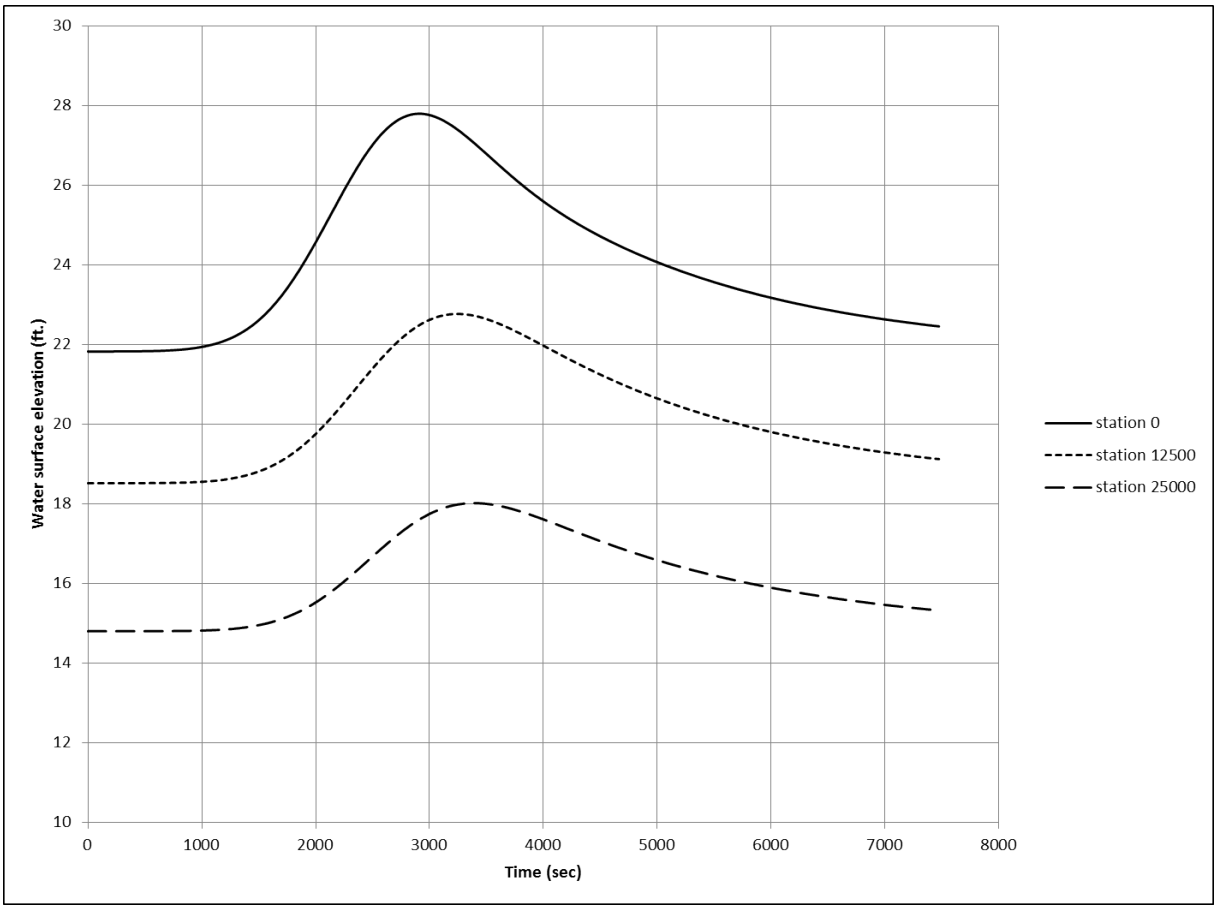


Figure 2.31: Water surface elevation at several station for $S = 2 \times 10^{-6}$ with the second hydrograph

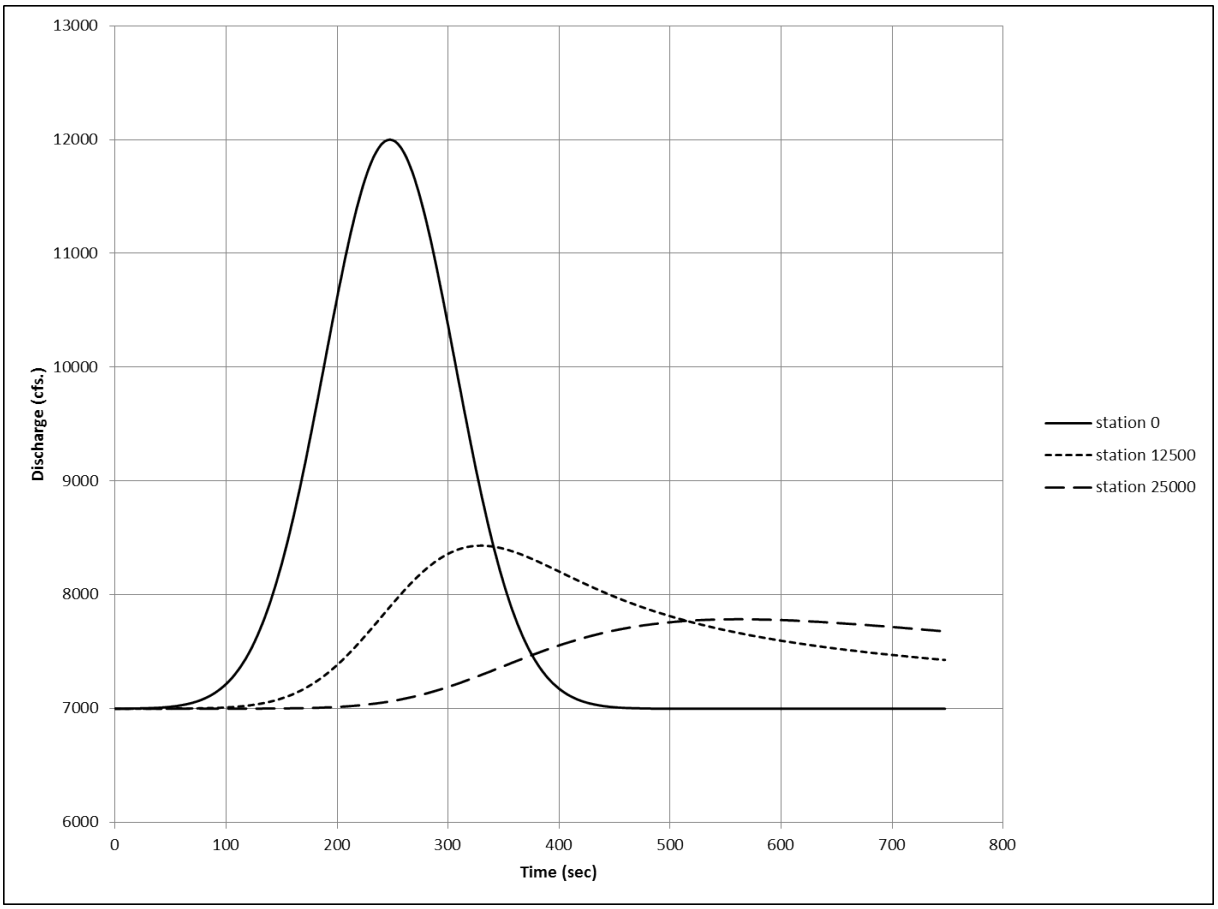


Figure 2.32: Discharge at several station for $S = 2 \times 10^{-5}$ with the first hydrograph

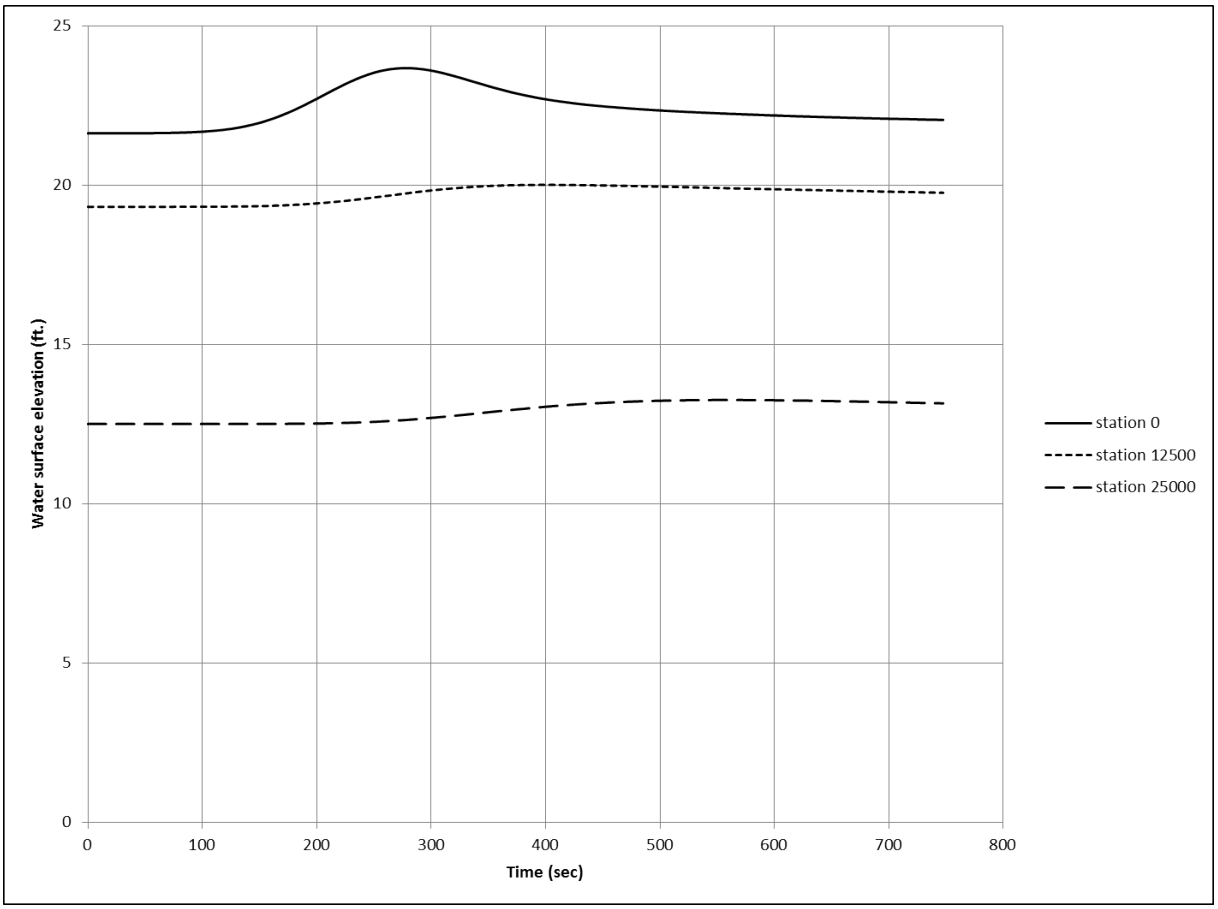


Figure 2.33: Water surface elevation at several station for $S = 2 \times 10^{-5}$ with the first hydrograph

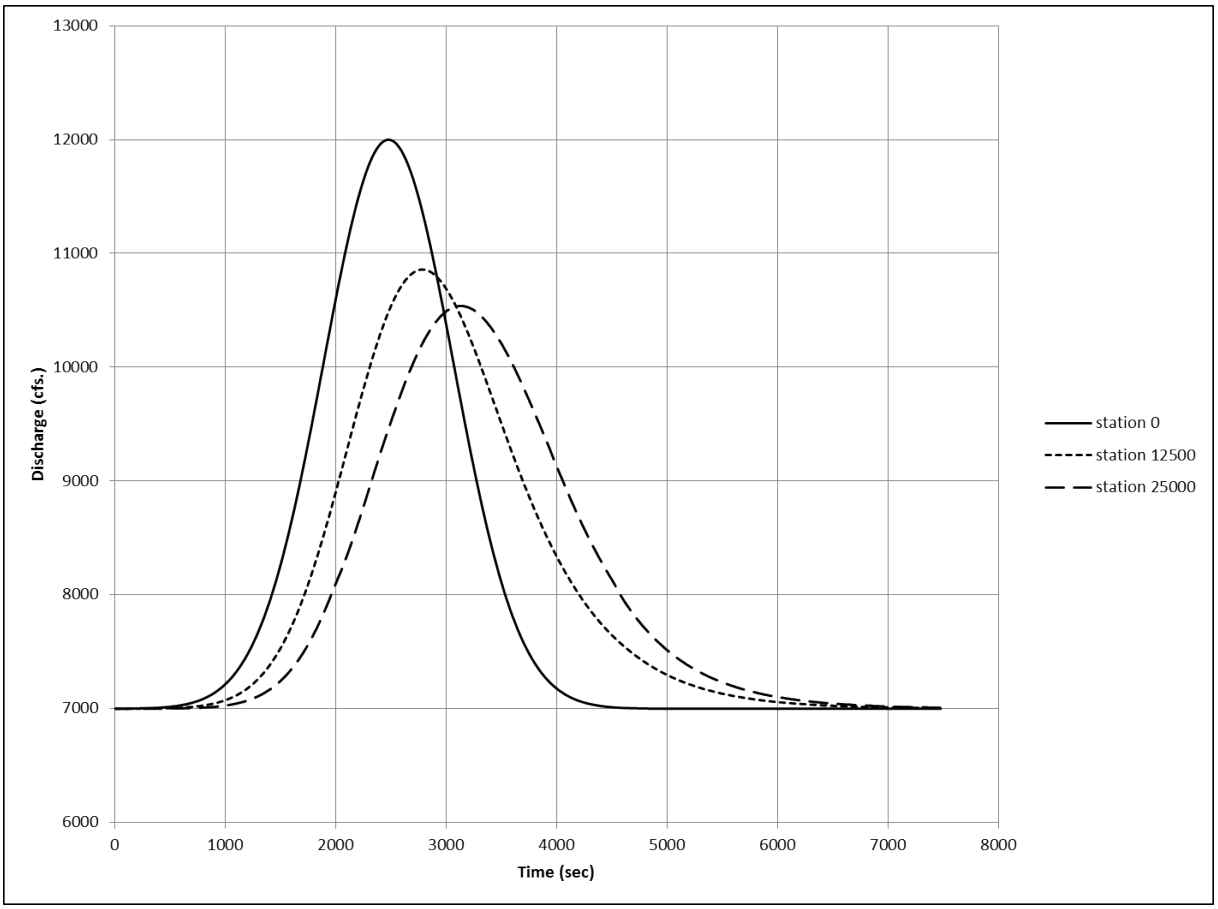


Figure 2.34: Discharge at several station for $S = 2 \times 10^{-5}$ with the second hydrograph

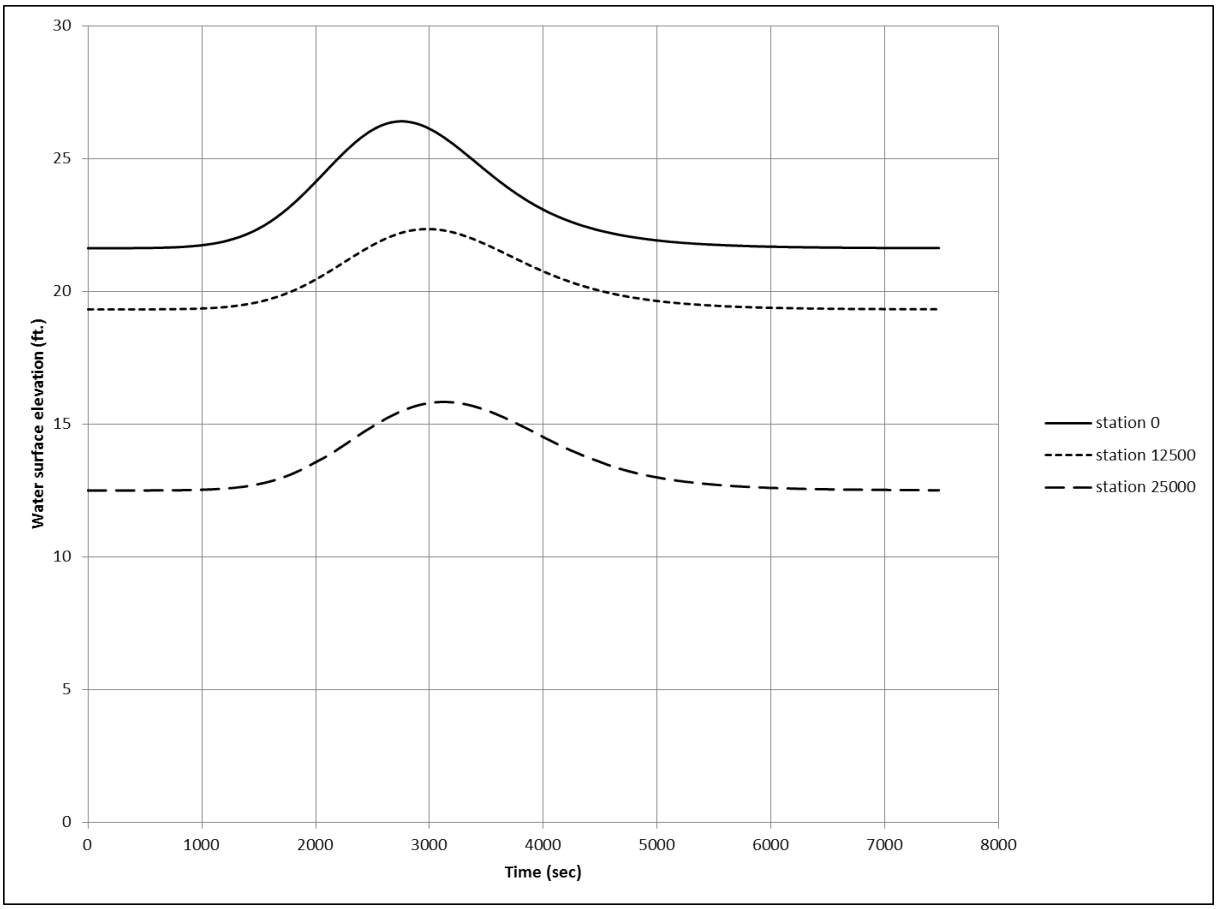


Figure 2.35: Water surface elevation at several station for $S = 2 \times 10^{-5}$ with the second hydrograph

Chapter 3

A SPACING CRITERIA

Here, the question of the accuracy of SRM is raised. Under the quasi-steady dynamic wave assumption, the convective acceleration term in $\frac{\partial Q}{\partial t}$ is neglected. And the system is discretized to allow a numeric solution. To make sure the result is reliable, it is necessary to check if this assumption is valid, in most cases it is valid and make sure that the spacing and the time step are chosen adequately.

It has been proven that (Gonzles Castro, J.A. and Ben Chie Yen , 2000 p74-76) [**?**] a method based on the quasi-steady dynamic wave assumption using interpolations is more accurate than the Non inertia wave method, more robust and less sensitive with respect to discretization.

The chosen time step also affects the accuracy of the model. A time step that is too big might not allow convergence. In the methodology of SRM, this is taken into account. (Zimmer Andrea 2013, p. 42) [**?**]. If, at a time step, the conditions do not allow convergence for Newton-Raphson method, the time step is divided by two and the values of the inflow for this new time step.

Even when all the previous conditions are met, the results might sometimes not be very accurate or even irrelevant. Some oscillations might occur. These problems may be linked to another condition.

The spacing is another element that must affect the accuracy of the method. It has to be checked at two stages: the step for the HPG/VPG creation and the length of conduit/reach chosen for each HPG/VPG. The HPG/VPG accounts for the variation in water surface elevation. Even though the HPG is made for a long reach, the method used to calculate might use an appropriate spacing. Thus the calculation of the water surface elevation might be accurate. However, none of the HPGs and VPGs can account for the change in discharge in the case of unsteady flow. If the variation in discharge with

respect to the longitudinal coordinate is important, the length of portion to make a HPG/VPG might not be accurate enough.

3.1 For steady flow (backwater length)

For steady subcritical flow, the length of the channel affected by a work downstream is called the backwater length. In the case of steady flow, the backwater length can be calculated with the following formula (Samuels, P. G. 1989) [**?**]:

$$L_b = 0.7\frac{D}{S_0} \tag{3.1}$$

With $D = \frac{A}{B}$. Or, for larger Froude Numbers:

$$L_b = 0.7\left(1 + {F_r}^2\right)\frac{D}{S_0} \tag{3.2}$$

For a steady flow calculation, in order to reduce the error, the length of the backwater from (3.2) has to include at least five spatial steps. The maximum spacing has to be:

$$\Delta x = 0.15\left(1 + {F_r}^2\right)\frac{D}{S_0} \tag{3.3}$$

This relationship was established assuming a trapezoidal geometry. Even if the channel is not trapezoidal, it gives a good idea of the backwater length and of the spacing criteria needed for hydraulic calculations.

3.1.1 Derivation

The derivation to find the backwater length is done using (1.3) introducing small perturbation to normal: $h = h_n + \eta$, (Samuels, P. G. 1989) [**?**], it gives the following results:

$$\frac{\partial (h_n + \eta)}{\partial x} = \frac{S_0 - S_f}{cos(\theta) - {F_r}^2} + \eta \frac{\partial}{\partial h} \left(\frac{S_0 - S_f}{cos(\theta) - {F_r}^2} \right)$$

$$\frac{\partial \eta}{\partial x} = \eta \left((S_0 - S_f) \frac{\partial}{\partial h} \left(\frac{1}{cos(\theta) - {F_r}^2} \right) + \frac{1}{cos(\theta) - {F_r}^2} \frac{\partial}{\partial h} (S_0 - S_f) \right) \tag{3.4}$$

The regime is close to normal: $S_0 - S_f \approx 0$. The derivative of the friction slope is (Samuels, P. G. 1989, p 575) [**?**]:

$$\frac{\partial S_f}{\partial h} = -S_f \left(\frac{10}{3R} \frac{\partial R}{\partial h} + \frac{2}{P} \frac{\partial P}{\partial h} \right)$$

Based on a typical cross-section geometry of a river channel (Samuels, P. G. 1989, p 575) [**?**] $\frac{dP}{dy}$ and $\frac{dR}{dy}$ is approximately equal to 1, and R is approximately equal to D Therefore, (3.4) becomes:

$$\frac{\partial \eta}{\partial x} = \eta \frac{10}{3D} \frac{S_f}{cos(\theta) - {F_r}^2} \tag{3.5}$$

The following expression is obtained from (3.5):

$$\eta = \eta_0 exp \left(-3.3 (x_0 - x) \frac{S_0}{D} \right) \tag{3.6}$$

The backwater length being defined as the distance where $\eta = 0.1\eta_0$, it has the following value:

$$L = 0.7 \frac{D}{S_0} \tag{3.7}$$

And, for larger Froude Numbers:

$$L = 0.7\left(1 - {F_r}^2\right)\frac{D}{S_0} \tag{3.8}$$

To obtain accurate results it might be useful to have at least 5 spacing contained in the backwater length:

$$L = 0.15\frac{D}{S_0} \tag{3.9}$$

3.2 Another spacing criterion that applies for unsteady flow

In the case of unsteady flow, a similar derivation can be made to find the length of the channel affected by unsteadiness. It provides the following formula:

$$L_u = 2.3 \frac{1}{\frac{\partial}{\partial Q}\left(\frac{\partial A}{\partial t}\right)} \tag{3.10}$$

The spacing criteria linked to the unsteadiness has to be proportional to this derivative.

$$\Delta x \sim \frac{1}{\frac{\partial}{\partial Q}\left(\frac{\partial A}{\partial t}\right)} \tag{3.11}$$

3.2.1 Derivation

In the case of quasi-unsteady flow, the first equation (conservation of momentum) of Saint-Venant is used under the assumption of steady flow. The unsteadiness is given by the second equation (conservation of mass):(1.2b). A derivation inspired by the one made by Samuels can be done, introducing a small pertubation to normal and applying it to the discharge:$Q = Q_n + \zeta$:

$$\frac{\partial \left(Q_n + \zeta\right)}{\partial x} = -\frac{\partial A}{\partial t} - \zeta \frac{\partial}{\partial Q}\left(\frac{\partial A}{\partial t}\right)$$

The differential equation can be obtained:

$$\frac{\partial \zeta}{\partial x} = -\zeta \frac{\partial}{\partial Q}\left(\frac{\partial A}{\partial t}\right) \tag{3.12}$$

and solved:

$$\zeta = \zeta_0 exp\left(-F\left(x\right)\right) \tag{3.13}$$

Where:

$$F\left(x\right) = \int_{x_0}^{x} \frac{\partial}{\partial Q}\left(\frac{\partial A}{\partial t}\right) \tag{3.14}$$

To simplify the derivation, $\frac{\partial}{\partial Q}\left(\frac{\partial A}{\partial t}\right)$ is assumed to be constant, (3.13) becomes:

$$\zeta = \zeta_0 exp\left(\frac{\partial}{\partial Q}\left(\frac{\partial A}{\partial t}\right)\left(x - x_0\right)\right) \tag{3.15}$$

Similarly to the backwater length, the length of unsteadiness can be defined as the distance where $\zeta = 0.1\zeta_0$. Therefore:

$$L_u = 2.3\frac{1}{\frac{\partial}{\partial Q}\left(\frac{\partial A}{\partial t}\right)}$$

3.3 Verification of unsteady flow spacing criteria

To check if this formula is valid, the simulations done previously will be done with different spacings. The results will be compared and the error can be calculated assuming the results will the smaller spacing is the reference.
The simulation of Wallingford is done with only 20 spacings, i.e. $\Delta x = 56.8ft.$. The theoretical case form Hoy's Thesis is done with 50 and 25 spacings instead of 100. And all the simulations for other theoretical cases are done for spacings equal to $\Delta x = \{500, 1250, 2500, 5000m\}$. In all cases, the error is calculated compared to the previously made simulations (the ones with the smallest spacings).

$$\epsilon = \frac{\overline{|Q_{firstsimulation} - Q_{\Delta x}|}}{Q_{firstsimulation}}$$

An SRM simulation gives a solution that has the form:

$$\left(Q_{i,j}, y_{i,j}, V_{\hat{i},j}\right)$$

The discharge $Q_{i,j}$ and the water surface $y_{i,j}$ elevation are given for each time step j and at each location i and the storage $V_{\hat{i},j}$ is given for each time step j and for each reach/subconduit $\hat{i}$.
The average discharge for each reach/subconduit is:

$$Q_{\hat{i},j} = \frac{1}{2}\left(Q_{i+1,j} + Q_{i,j}\right)$$

Using (3.11), and noticing that $Q = A \times \Delta x$, the ratio of the spacing over the spacing criterion α can be calculated:

$$\alpha = \frac{\Delta x}{\Delta x_0} = \frac{\partial}{\partial Q}\left(\frac{\partial V}{\partial t}\right) \tag{3.16}$$

This equation can be discretized and applied to the results given by SRM to check if the chosen spacing is fine.

$$\left(\frac{\Delta x}{\Delta x_0}\right)_{\hat{i},j} = \frac{1}{Q_{\hat{i},j+1} - Q_{\hat{i},j}} \left(\frac{V_{\hat{i},j+1} - V_{\hat{i},j}}{\Delta t} - \frac{V_{\hat{i},j} - V_{\hat{i},j-1}}{\Delta t}\right) \tag{3.17}$$

The ratio is calculated for each point and each time step.

3.3.1 Graphs showing error vs. spacing/unsteadiness criteria

This error is plotted versus how often the ration given by (3.17) is higher than 1.

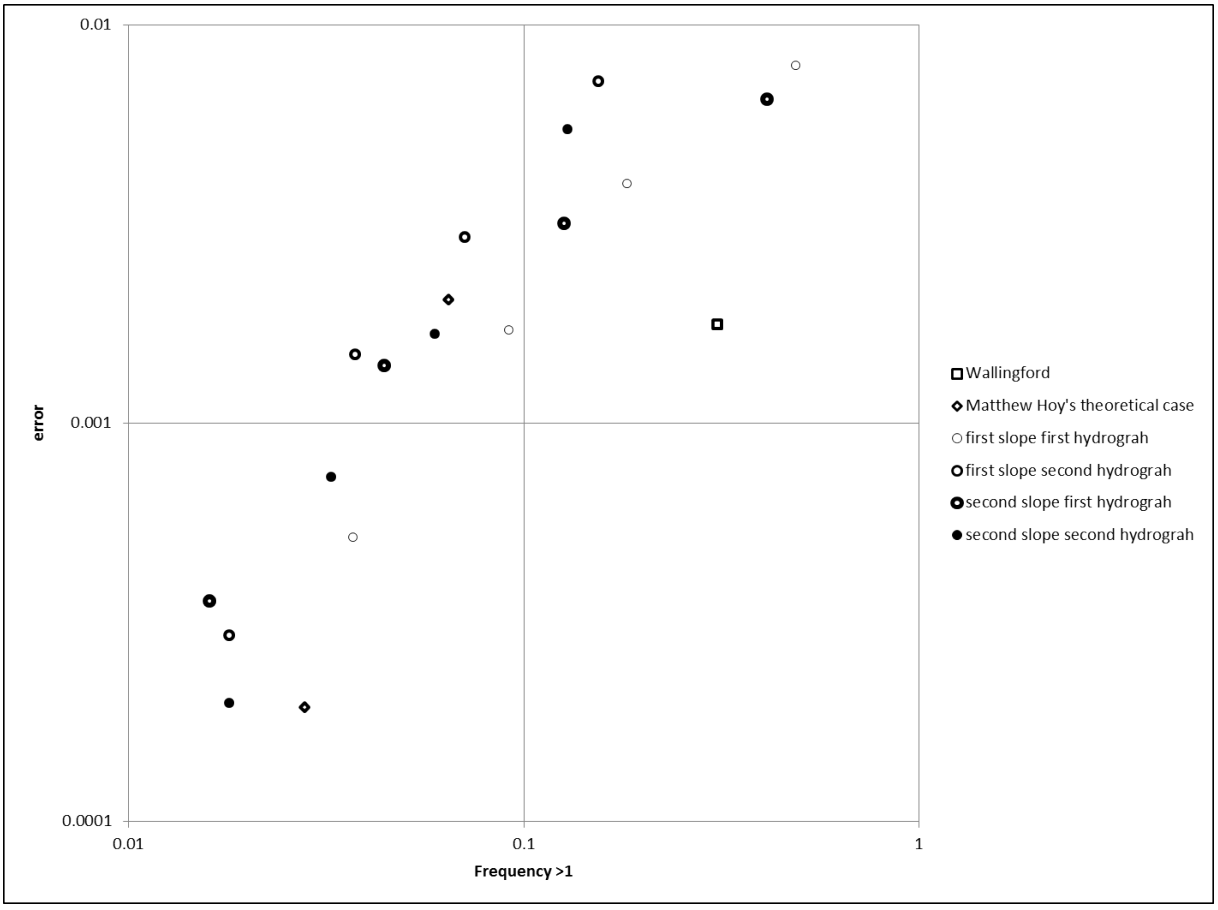


Figure 3.1: the error vs the frequency of ratio higher than 1

The error seems to be related to the frequency of points (location and time) where the ratio os higher than 1. However it stays lower than 0.01 even if

the frequency is close to 1. Therefore, it must be interesting to see how it can be related to the frequency of points where the ratio os higher than other numbers.

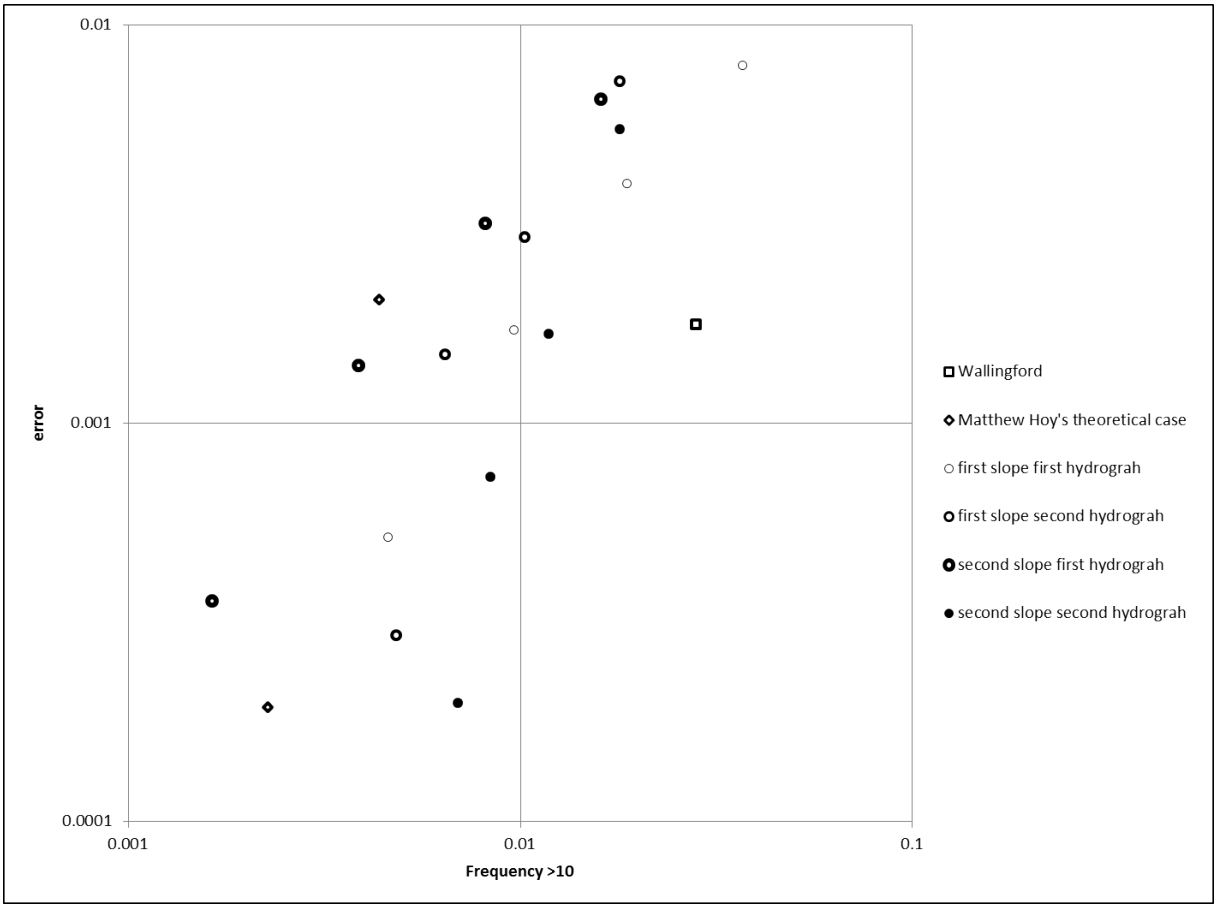


Figure 3.2: the error vs the frequency of ratio higher than 10

In the case of 10, the same observations as in the case of 1 can be done. Even if the points are more spread out in the case of 100, the graph shows that the order of magnitude of the error and of the frequency are the same. In the cases tested, the accuracy of the result is good. It should be interesting to see what happens when the oscillations start occurring and the accuracy .

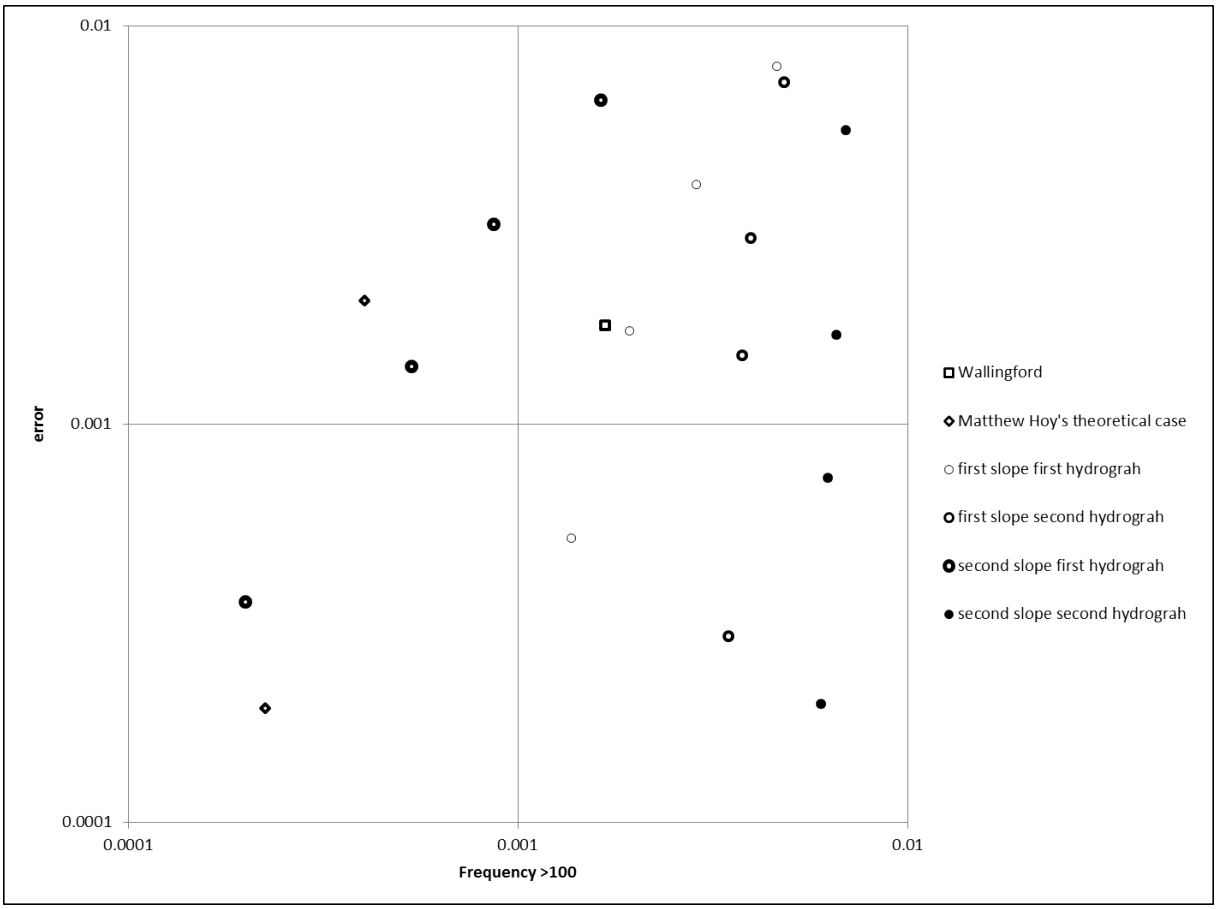


Figure 3.3: the error vs the frequency of ratio higher than 100

3.3.2 spacing criteria that causes oscillations

Wallingford Hydraulic research Station

In the case of Wallingford Hydraulic Research Station, for a spacing of $\Delta x = 113.6 ft.$ the accuracy decreases and the oscillations start occurring:

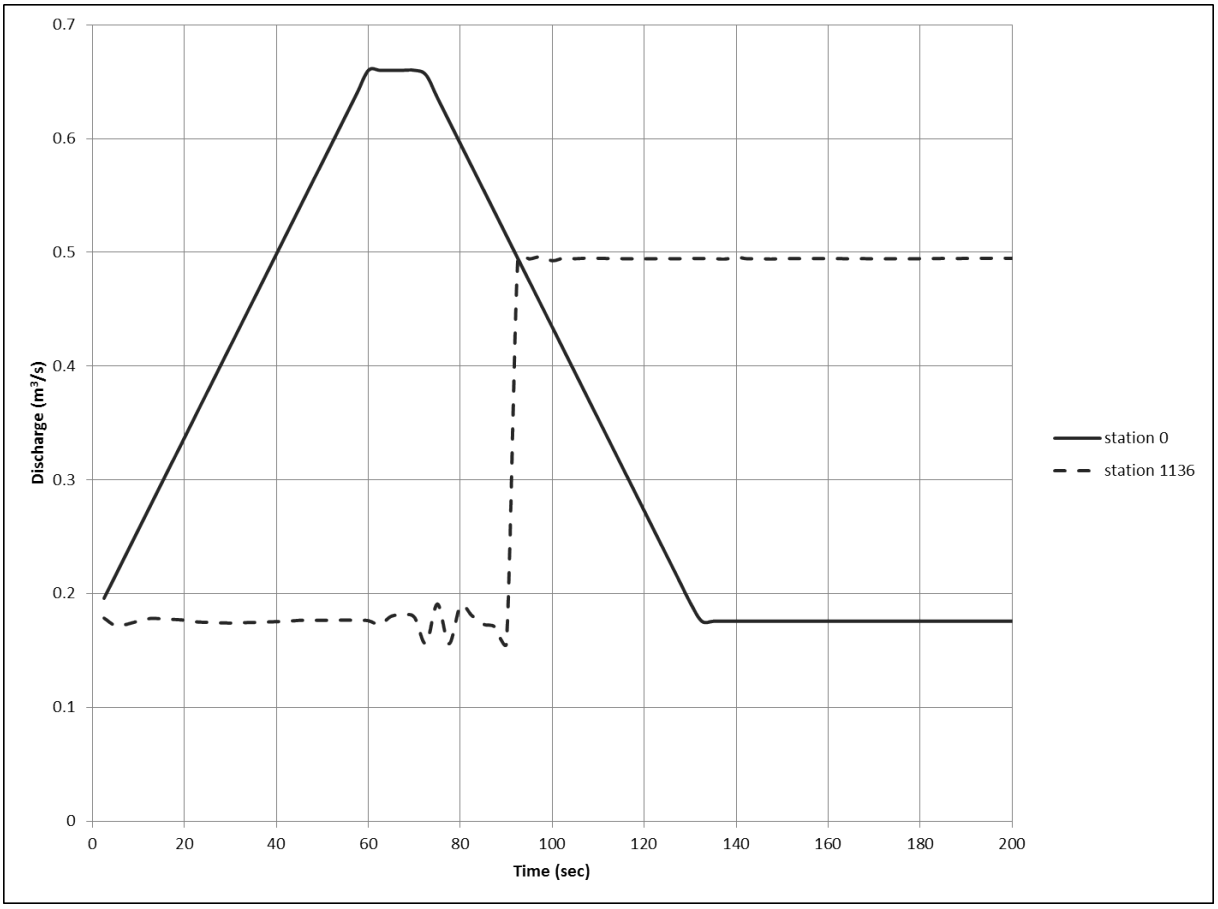


Figure 3.4: results for the discharge with a spacing of 113.6 ft

First theoretical case

In the first theoretical case described previously, for a spacing of $\Delta x = 200 ft$. the accuracy decreases and the oscillations start occurring:

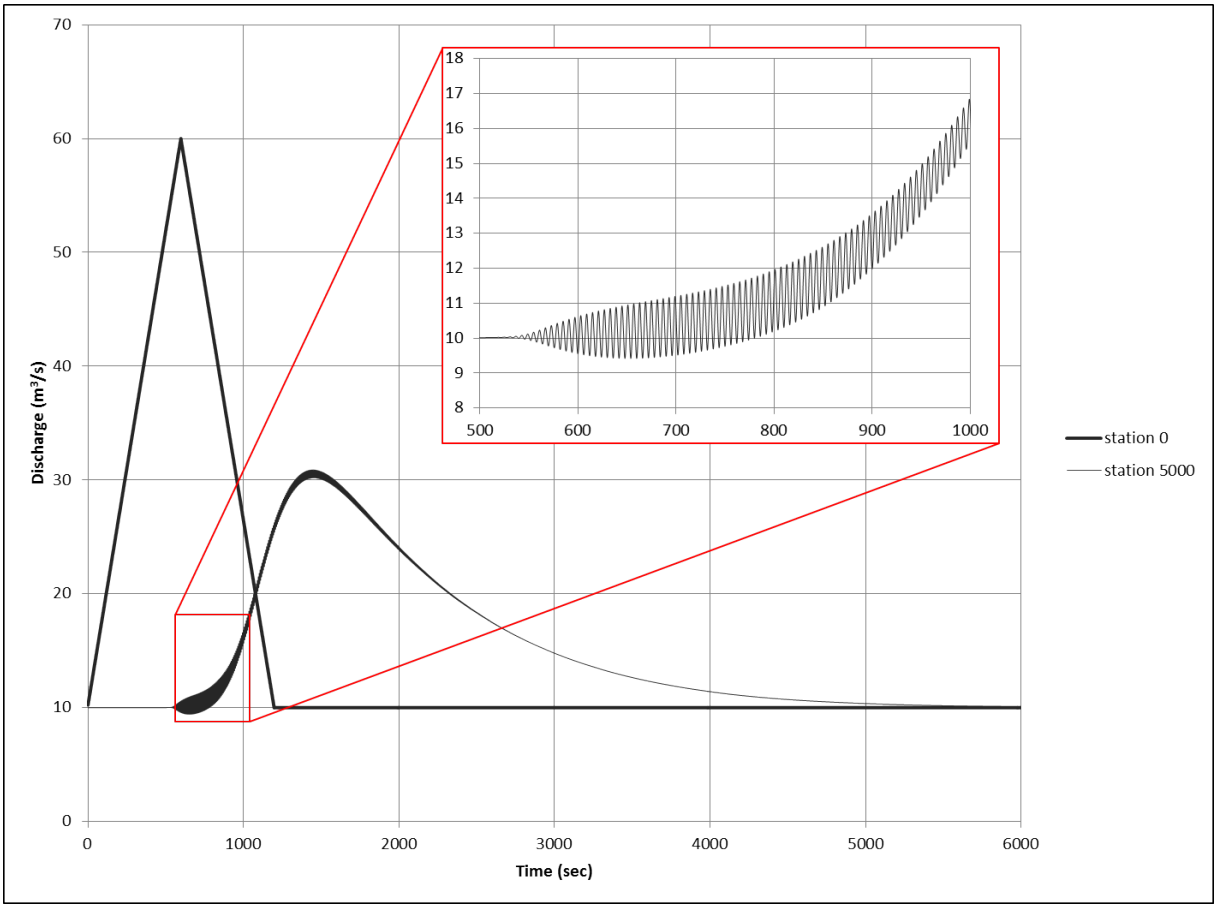


Figure 3.5: results for the discharge with a spacing of 50 ft

First theoretical case another spacing

When the spacing is changed to $\Delta x = 6250ft.$ the error and the oscillations become more important and the results is not relevant anymore:

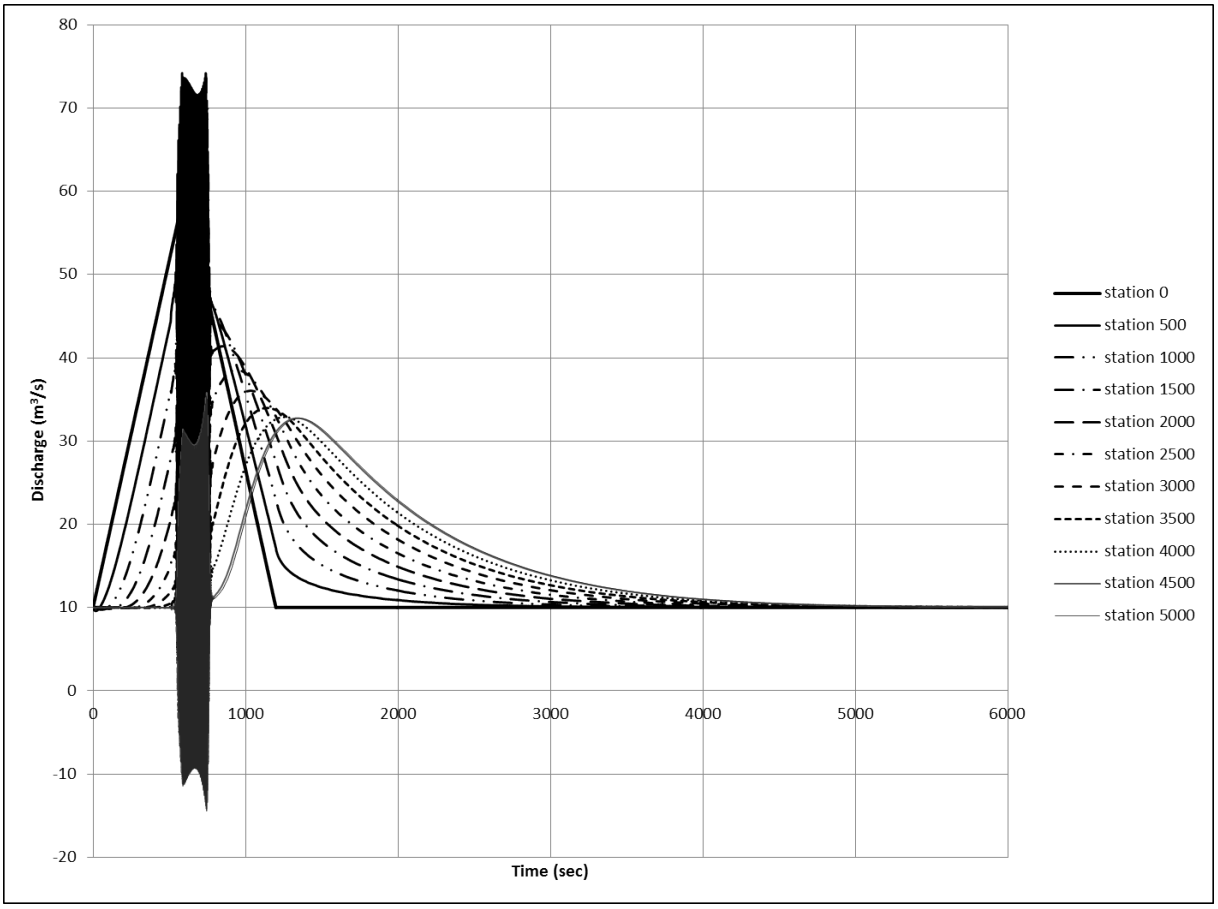


Figure 3.6: results for the discharge with a spacing of 50 m

Other theoretical cases

For all other theoretical cases, the oscillations and the lack of accuracy in the results starts appearing for a spacing of $\Delta x = 6250ft$. the error and the oscillations become more important and the results is not relevant anymore:

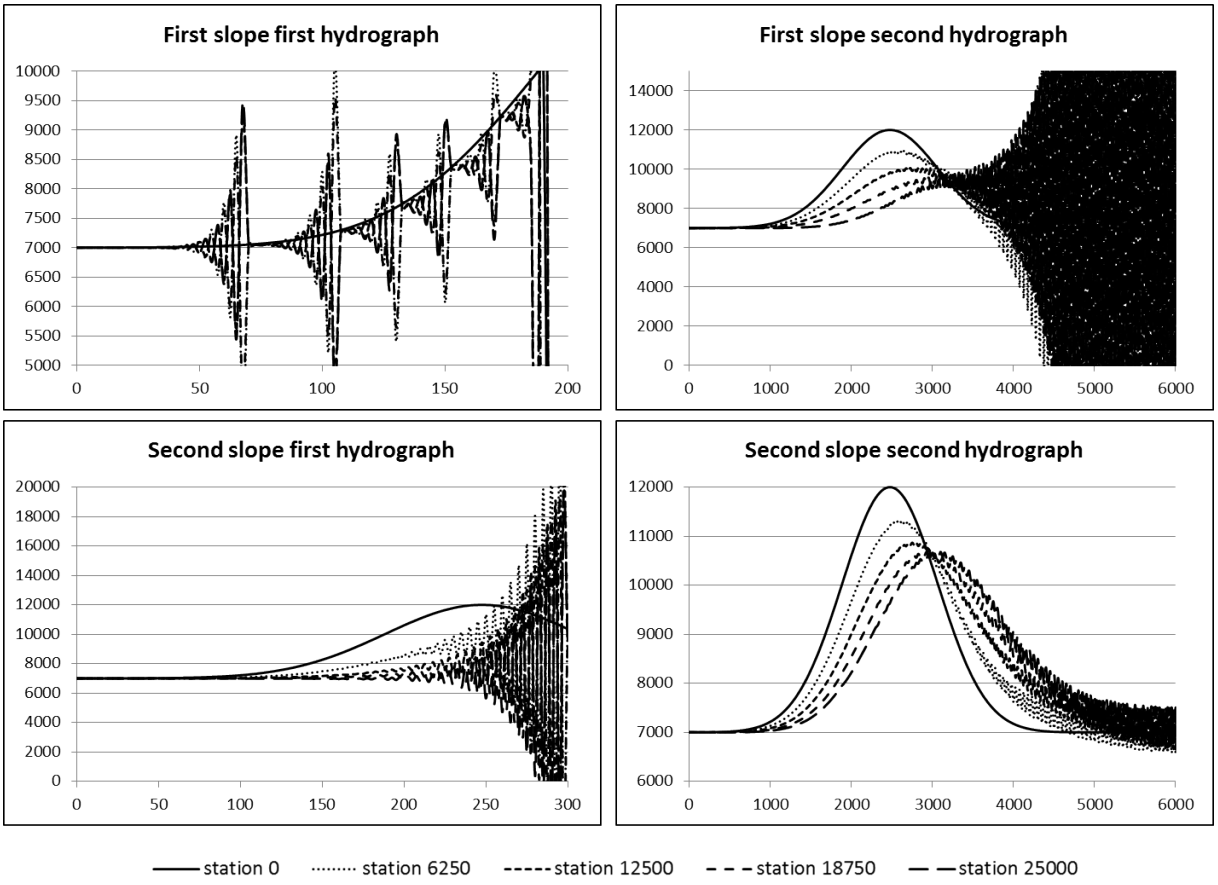


Figure 3.7: results for the discharge with a spacing of 6250 ft

3.3.3 Other theoretical cases

For all cases, it is interesting to see the values of the ratio α when the lack of accuracy occurs.
It has a different value for all locations and time steps, therefore, the distributions is plotted:

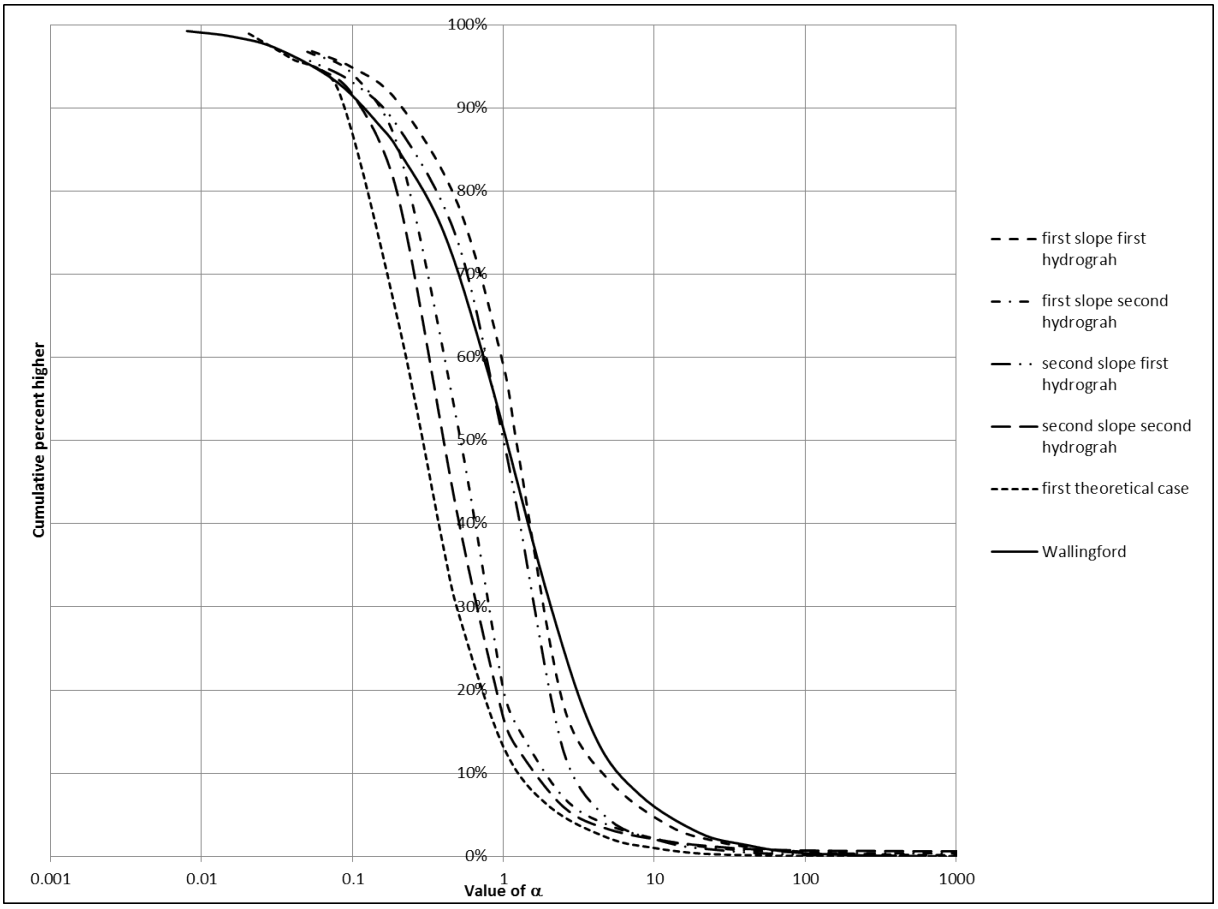


Figure 3.8: distribution of the ratio α for all cases at the "limit" spacing

In all cases, for more than 10 % of the points (time step and locations), the ratio α is higher than 1. It can be deduced that it is a condition to obtain a satisfying result.

Chapter 4

CONCLUSION

The storage Routing Method is a method of hydraulic calculation based on Saint-Venant equations that can be accurate and efficient under the following assumption: It applies under the following assumptions:

- The flow is 1-dimensional, only the variations in the flow wise direction are considered.
- The pressure is assumed to be hydrostatic in the entire channel, the vertical acceleration are neglected
- The channel can be considered as a straight line
- The geometry of the channel is fixed (no effect of scour and deposition)
- The slope is small
- The fluid is incompressible
- the Quasi-steady flow assumption: $\frac{\partial Q}{\partial t}$ is negligible.
- The spacing used to create the HPGs/VPGs respects the spacing criterion linked to the backwater length.
- The time step is small enough to allow convergence
- The spacing used to divide the conduits/reaches in SRM respects the criterion linked to unsteadiness.

The assumption to apply Saint-Venant equations and the Quasi-steady wave assumption have to be verified to use SRM, otherwise, the method of calculation has to be changed. The model used for SRM calculations can be

adapted to meet the time step and spacing criteria.
The time step criterion is already taken into account (Zimmer, A. et al., 2013) [**?**] since the code is dividing it until it finds convergence. The spacing criterion linked to the backwater length has no effect on the spacing taken for SRM. It has to be taken into account to create the HPGs/VPGs.
The HPGs/VPGs do not account for the change of discharge. Therefore the spacing criterion linked to unsteadiness is important here. The spacing for SRM has to be chosen first to meet this criterion.

Suggestions for future work

The SRM code implemented with matlab does not account for the spacing criterion linked to unsteadiness. This can be implemented by doing HPGs/VPGs for different sizes of reaches/conduits and the code will choose the spacing based on the criterion calculation at each time step.
The second thing that can be improved is the method of convergence used by SRM. Newton-Raphson is efficient, however, there are cases for which it does not work. Another method of convergence (maybe less efficient but which might work for all cases) when Newton-Raphson does not converge.
The method to create HPGs/VPGs used by SRM can also be improved. It only works for circular conduits. in the case of a more complicated geometry, it is necessary to create the HPGs/VPGs with another method and use them as input to the code. It may be convenient to have a code that takes many types of geometry into account (other shapes and non-prismatic channels).

Appendix A

USER'S MANUAL

A.1 Introduction

Irregular precipitations as well as irregular use of water cause the flow in rivers and hydraulic infrastructure to be unsteady and non-uniform. During critical events (heavy rainstorm for example), it is useful to know some estimations of the discharge and the water surface elevation, it must helps preventing overflows and floods. A typical way to obtain these values is to use the Saint-Venant equations and solve them numerically. These equations are non-linear and quite complex. Thus is it often useful to make some assumptions to neglect some terms and simplify the calculation. It may allow simpler computation and faster results without losing to much accuracy. SRM (storage routing method) is a matlab code that applies this method under the quasi-steady dynamic wave assumption. It has been developed and enhanced by previous authors' work in order to "create a method for modeling these unsteady flows in a straightforward and efficient manner" (Matthew A. Hoy, 2005) [**?**].
The methods of calculation based on Saint-Venant apply for a one-dimensional flow, an incompressible fluid, with a hydrostatic pressure, no scour or deposition and a small slope. These equations being non-linear, no general analytical solution exists. In the case of unsteady flow, no term is neglected, the computation can be intensive. On the other hand, when too many terms are neglected, the result might not be accurate enough. In some cases, the convective acceleration can be neglected and the quasi-steady dynamic wave is a good compromise between accuracy and computation velocity. SRM is one method of calculation based on the quasi-steady dynamic wave assumption. It creates table of values of upstream water surface elevation (HPGs) and

volume of water (VPGs) depending on the discharge and the downstream water surface elevation for all portions of the system considered beforehand and uses interpolations of these graphs at each time step of the calculation to find the solution.

A.2 Description of the code

SRM uses SWMM files (describe SWMM) to read the properties of each component of the system or a river (reach, pipe, junction, weir, etc.). It either creates or uses input HPGs/VPGS for each river reach/pipe. With given boundary conditions (inflow/outflow hydrographs, stage hydrographs, rating curves) it calculates the discharge, the water surface elevation and the storage everywhere in this system. The calculation is made using interpolation of HPGs/VPG and the method of convergence used at each time step is Newton-Raphson.

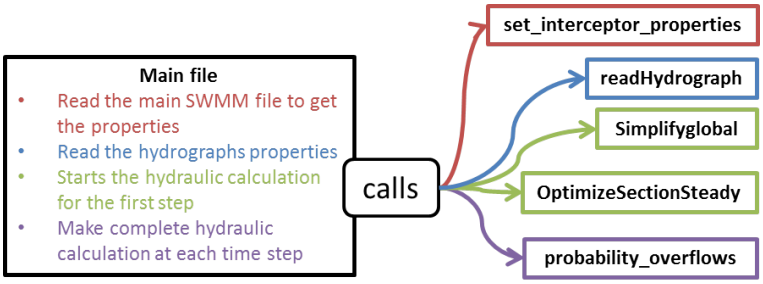


Figure A.1: overview of the code

A Read the inputs

The geometry

The function "set_interceptor_properties" reads the SWMM file as a text file, line by line, to get the inflow hydrographs, and convert all the component of the system in a global structure variable called *interceptor*.

Reading each element

The outlets, conduits, orifices, outfalls, pumps, weirs and storage units are saved in the global variable *interceptor* with their geometric properties and

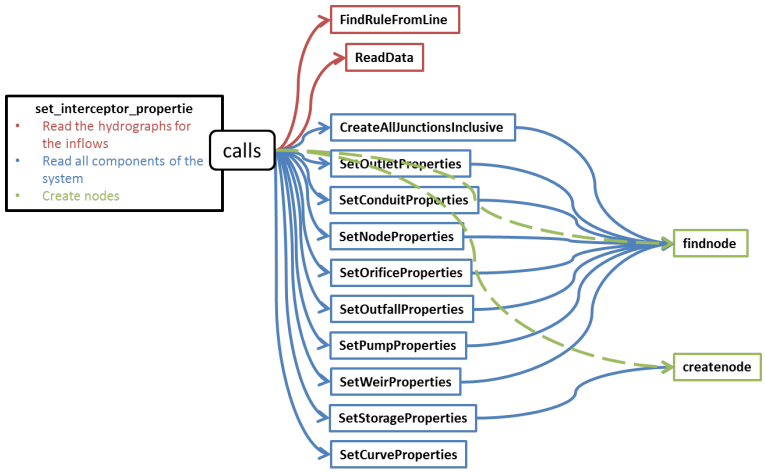


Figure A.2: set the global variable *interceptor*

functions between at least two variables (water surface elevations, storage, discharge). all this elements are considered as links in SRM.

The junctions

The junctions in SWMM become nodes or junctions in SRM. A junction that joins two links only (for example two conduits) becomes a node. A junction that joins more than two links or an inflow is saved as a junction in SRM and nodes are created for each link and inflow, see Figure A.3. Each junctions is saved with the properties of the nodes it contains.

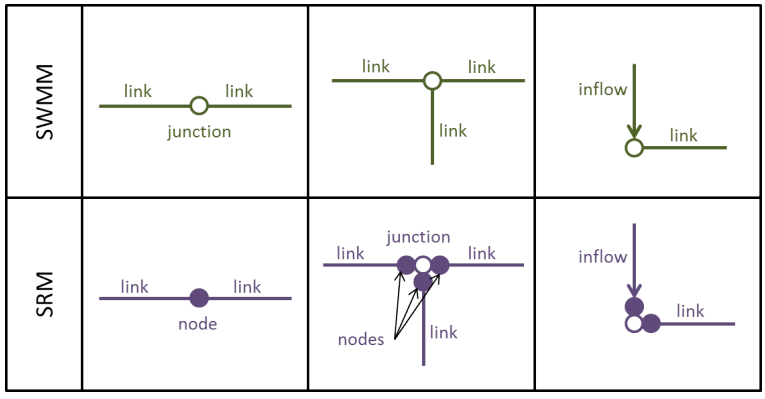


Figure A.3: The junctions in SRM

The conduits

In the case of the conduits, there are two functions making links between variables: the HPG (the upstream water surface elevation as a function of the downstream water surface elevation and the discharge) and the VPG (the volume of water as a function of the downstream water surface elevation and the discharge). SRM contains a function to create the HPGs/VPGs given the properties of the conduits. This function assumes a circular geometry. In the case of another shape or a river reach, it is possible to create the HPGs/VPGs before running the code and use them as an input.

The hydrographs

The function "set_interceptor_properties" calls subfubction to read the time series of the SWMM files. This works only for inflow hydrographs. If other solutions are needed, it is possible to use the function "readHydrograph", to creates new variables for the inflow/outflow/stage hydrographs and use them in the next parts of the code for the calculation.

B Initialize the calculation

The function "Simplifyglobal" calculate the discharge at each node of the system for steady flow using the first value of the given hydrograph(s).
The values of the dicharge in all the nodes are used in the function "OptimizeSectionSteady" as a first guess for the Newton-Raphson method for convergence for steady flow.

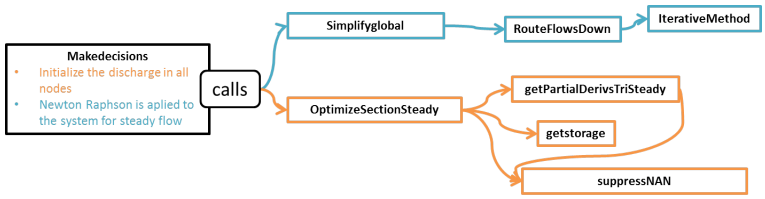


Figure A.4: initialization: steady flow calculation

The equations for Newton-Raphson

At each node, there are two unknowns: the discharge and the water surface elevation.
Each link gives two equations, one is linked to the conservation of mass, the other one to the momentum equation.
Each junctions gives as many equations as it has nodes.

Therefore, this becomes a system of unknowns and equations.
There are usually more unknowns than equations. To complete the number of equations, boundary conditions are necessary.

The equations are (Zimmer, A. et al., 2013, p947) [?]:

$$F = (f_i)$$

- for junctions:

$$\begin{cases} f = y_i - y_j & \text{if } i \neq j \\ f = \sum Q_{in} - \sum Q_{out} \end{cases}$$

- for links:

$$\begin{cases} f = Q_{in} - Q_{out} & \text{conservation equation} \\ f = y_{up} - f\left(y_{down}, Q\right) & \text{momentum equation} \end{cases}$$

- Special case of conduits/reaches

$$\begin{cases} f = Q_{in} - Q_{out} \\ f = y_{up} - HPG\left(y_{down}, Q\right) \end{cases}$$

- The boundary conditions

$$f = \begin{cases} Q - inflow/outflow & \text{inflow or outflow hydrograph} \\ f = y - stage & \text{stage hydrograph} \\ f = rating\left(y, Q\right) & \text{rating curve} \end{cases}$$

The method used to find the roots of this system is Newton Raphson. The partial derivatives of these functions with respect to the discharges and the water surface elevations at every node need to be calculated:

$$\Delta F = \left(\frac{\partial f_i}{\partial x_j}\right)_{ij}$$

The iteration of Newton-Raphson method is:

$$F = F_{prev} + \left(\Delta F\right)^{-1} + \Delta X$$

Until F and F_{prev} are close enough.
The equations F for Newton-Raphson are set by the subfunction "FindFunctions" and the derivatives ΔF by the subfunction "FindJacobian".

C Unsteady flow calculation

The same type of calculation as in the previous part is done. Under the unsteady flow assumption, the equations are slightly changed(Zimmer, A. et al., 2013, p947) [**?**]:

$$F = (f_i)$$

- for junctions:

$$\begin{cases} f = y_i - y_j & \text{if } i \neq j \\ f = \sum Q_{in} - \sum Q_{out} \end{cases}$$

- for links:

$$\begin{cases} f = \Delta t \left(Q_{in} - Q_{out}\right) - \Delta volume & \text{conservation equation} \\ f = y_{up} - f\left(y_{down}, Q\right) & \text{momentum equation} \end{cases}$$

- Special case of conduits/reaches

$$\begin{cases} f = \Delta t \left(Q_{in} - Q_{out}\right) - \Delta VPG\left(\frac{Q_{in}+Q_{out}}{2}\right) \\ f = y_{up} - HPG\left(y_{down}, Q\right) \end{cases}$$

- The boundary conditions

$$\begin{cases} f = Q - inflow/outflow & \text{inflow or outflow hydrograph} \\ f = y - stage & \text{stage hydrograph} \\ f = rating\left(y, Q\right) & \text{rating curve} \end{cases}$$

The method of convergence used here is also Newton-Raphson.

In the case of unsteady flow, the time step has to be taken into account. It influences the convergence. When the time step is not precise enough, there might be no convergence. Therefore, the code contains a function that divides the time step in this case. The function “AdjustTimeStep” calls “Op-

timizeSection", the function that applies Newton-Raphson for unsteady flow. It check the convergence, it returns the values if it is good, if not, it divides the time step and calls itself twice. The time step can be divided endlessly, however,when the time step becomes too small, "OptimizeSectionSteady" is called to make the calculation for steady flow and return the values.

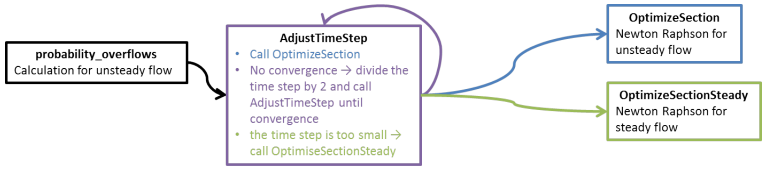

probability_overflows
Calculation for unsteady flow
AdjustTimeStep
Call OptimizeSection
No convergence → divide the time step by 2 and call AdjustTimeStep until convergence
the time step is too small → call OptimiseSectionSteady
OptimizeSection
Newton Raphson for unsteady flow
OptimizeSectionSteady
Newton Raphson for steady flow

Figure A.5: unsteady flow calculation

D The output of the code

The values of the discharge and the water surface elevation at all nodes and of the volume of water in all links are stored in the global variable *interceptor*: *interceptor.node(i).flow*, *interceptor.node(i).wse.* and *interceptor.conduit(i).storage* As well as in the variables *WSEM*, *flowM* and *storageM.*

A.3 How to use SRM

The Matlab code for SRM includes a function that is making HPGs and VPGs for circular pipes.
In this case, the input of the code has to be only a SWMM file.

A The calculation

The code is making the calculation using the timeseries from SWMM as inflow hydrographs and assuming free outfalls.

B How to set the input

The input has to be a SWMM file. All the properties of the system in this file will be used for the calculation.
The name of the SWMM has to be reported in the main file as *inputfile* "makedecisions" of the SRM code, line 6:

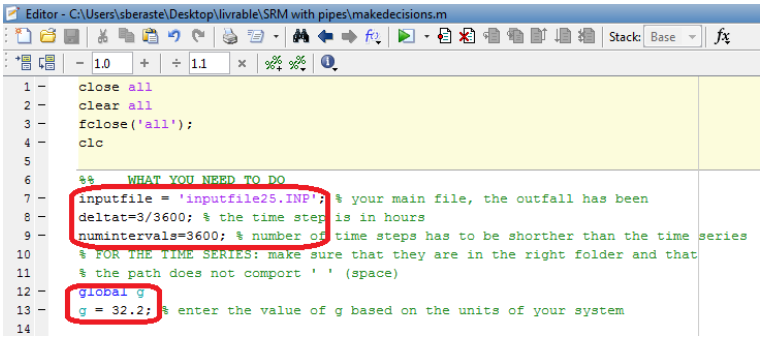

Editor - C:\Users\sberaste\Desktop\livrable\SRM with pipes\makedecisions.m
Stack: Base
1 close all
2 clear all
3 fclose('all');
4 clc
5
6 %% WHAT YOU NEED TO DO
7 inputfile = 'inputfile25.INP'; % your main file, the outfall has been
8 deltat=3/3600; % the time step is in hours
9 numintervals=3600; % number of time steps has to be shorther than the time series
10 % FOR THE TIME SERIES: make sure that they are in the right folder and that
11 % the path does not comport ' ' (space)
12 global g
13 g = 32.2; % enter the value of g based on the units of your system
14

Figure A.6: What to change i the main file *makedecisions*

It is also necessary to specify the number of time steps *numintervals* and the value of the time step *deltat* for the calculation, this can be done in the main file, "makedecisions" of the SRM code, line 7:

Once all this is set up, the simulation can be run and the progress will appear in the command window.

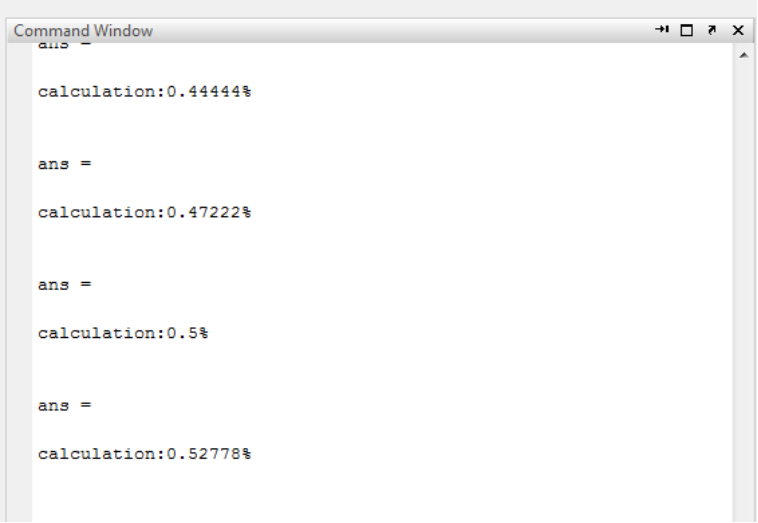


Figure A.7: progress of the calculation

C The calculation

As specified previously, the values of the discharge and the water surface elevation at all nodes and of the volume of water in all links are stored in the global variable *interceptor*: *interceptor.node(i).flow*, *interceptor.node(i).wse.* and *interceptor.conduit(i).storage* As well as in the variables *WSEM*, *flowM* and *storageM.*
Once the calculation is done, some graphs with (...) will appear. You can use them as reulsts, but you can also use the variables cited previously to make your own graphs or to get the numerical values.
It can be convenient for example to get the values in excel files, int his case, the commands:

- *xlswrite('flow.xlsx',flowM);*
- *xlswrite('wse.xlsx',WSEM);*
- *xlswrite('storage.xlsx',storageM);*

A.4 Alternatives for using SRM

By default, SRM makes the calculation for circular pipes and takes the inflow hydrographs and the free outfalls as boundary conditions. These options can be changed.

A Non-circular conduits/river reaches

In the case on non-circular conduits or river reaches, it is possible to create the HPGs/VPGs with another tool and use them as an input for SRM.

A tool to create HPGs/VPGs

An alternative to create the HPGs/VPGs is the HPG utility for HEC-RAS. It uses the files from HEC-RAS to create HPGS/VPGS. The manual *HPG Utility for HEC-RAS Users Manual* (Blake J. Landry and Nils Oberg, 2013) [**?**] explains how to use this tool.

How to use this tool

A matlab code has been cerated to make HPGs and VPGs for all reaches/conduits. In order to use it, it is first necessary to split the HEC-RAS river file in smaller river reaches files.
For example, in the case of the Chicago River, the river in the file comports 13 reaches. It has to be first separated in 13 files.

A reach can be too long to be a precise enough spacing for SRM, so it is sometimes necessary to divide it in smaller parts. See figure A.9.

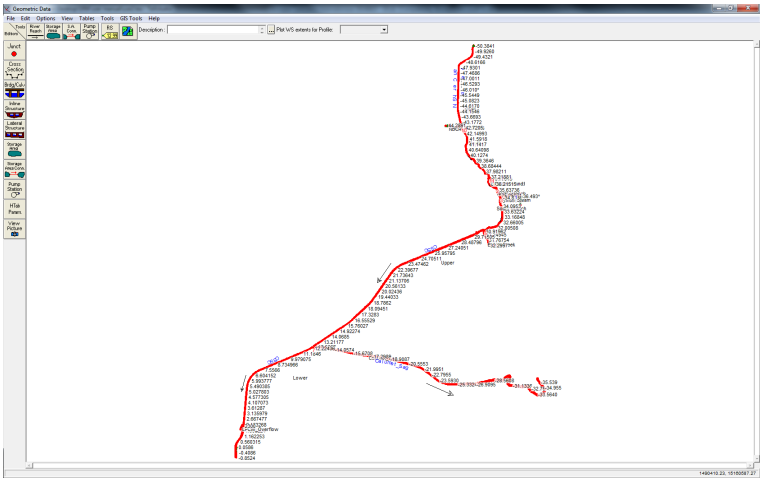


Figure A.8: HEC-RAS file for the Chicago River

Sometimes, the river can flow in both directions as it is the case for the Chicago River Main Stem. In this case it might be necessary to have all these files in both directions, see figure A.10. (HEC-RAS specifies a direction for the flow)

A list of the invert elevation of the cross-sections at each end of the parts is necessary.
They have to be classified in order from upstream to downstream. This list has to be in a text format and called *inverts*. This file can be made with excel, the first column is the station and the second is the invert elevation, see figure A.12.

Once all these files are created, they can be used to create HPGs and VPGs. They first have to be first classified in folders. The main folder has to contain the *inverts* file and two folders: *backward* and *forward*. These folders contain the HEC-RAS files made previously, the folder *forward* contains all the files were the river is flowing in the streamwise direction, the older *backward* the same files for the opposite direction. In both folders, all the HEC-RAS files are classified in numbered folders. The numbers go from upstream to downstream.

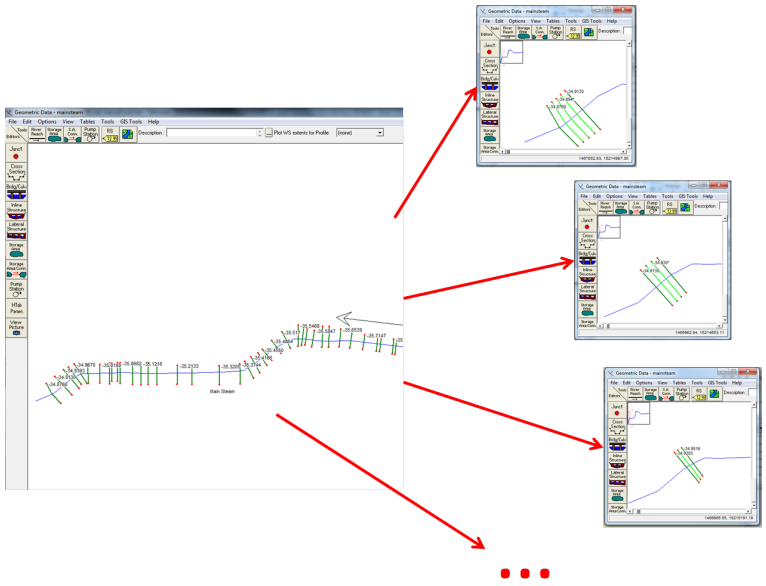


Figure A.9: Division of the Chicago River Main stem reach

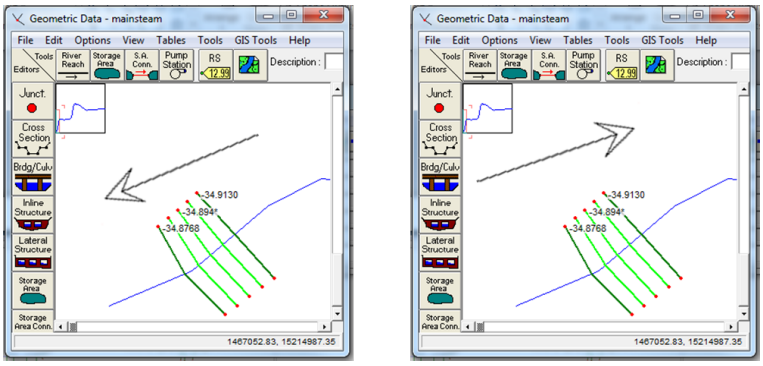


Figure A.10: Two possible directions for a reach

When everything is set, the code can be run. In the main folder, there should be the Matlab file *createHPG* once opened, the the number M and N have to be changed. M have to be the first number of the folders made before and N the last one. The maximum water surface elevation has also to be changed, see figure A.14.

The code to create HPGs doesnot always work perfectly, sometimes, HPG files rae not created, sometimes, the values found for the volume of water in the VPG are all equal to 0, in these cases, the HPG files have to be done

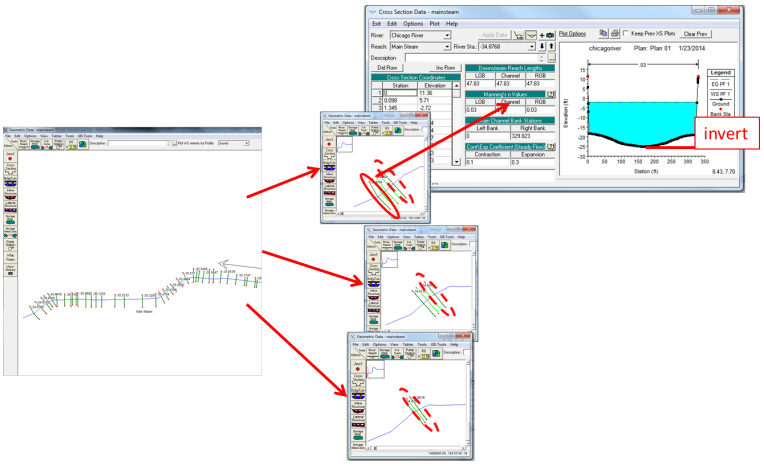
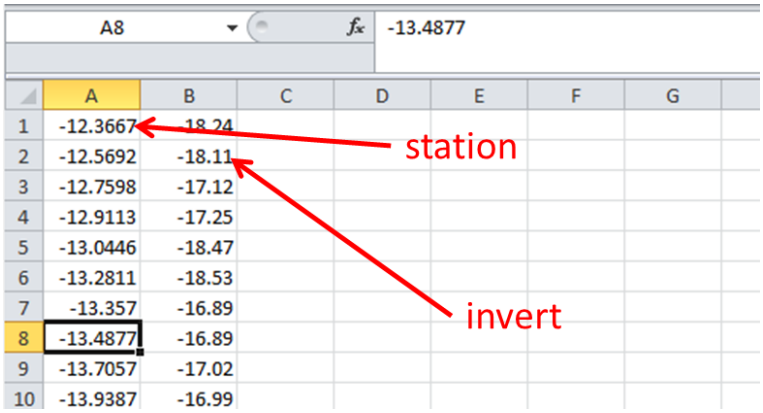


Figure A.11: where to find the invert elevation



	A	B	C	D	E	F	G
1	-12.3667	-18.24					
2	-12.5692	-18.11					
3	-12.7598	-17.12					
4	-12.9113	-17.25					
5	-13.0446	-18.47					
6	-13.2811	-18.53					
7	-13.357	-16.89					
8	-13.4877	-16.89					
9	-13.7057	-17.02					
10	-13.9387	-16.99					

Figure A.12: how to set the inverts file

manually. To verify that all HPGs and VPGs have been created and the values are correct, it is possible to run the matlab code *HPGexists*. If a file is missing or if it exist and the VPG comports 0 values, it will notify it, see figure A.15.

Once all the files are created and valid, the values can be stored in a Matlab structure file to be used in SRM. In this file too, it is necessary to set the values M and N.

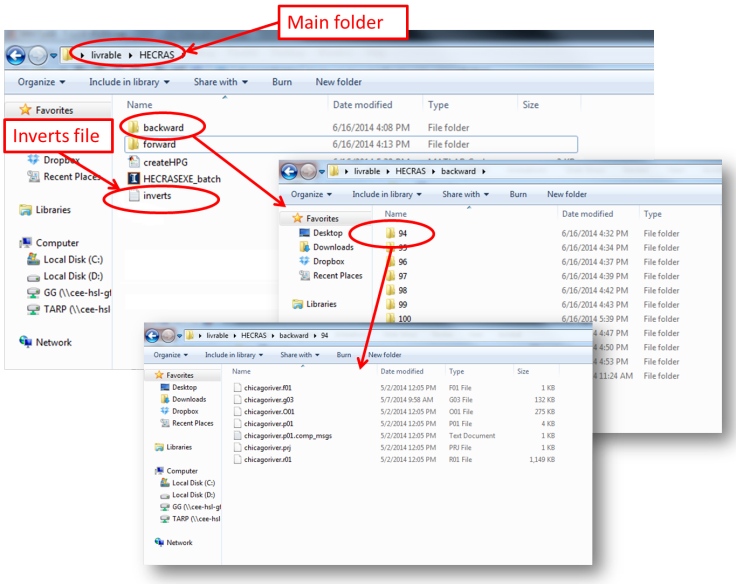


Figure A.13: How to classify the files

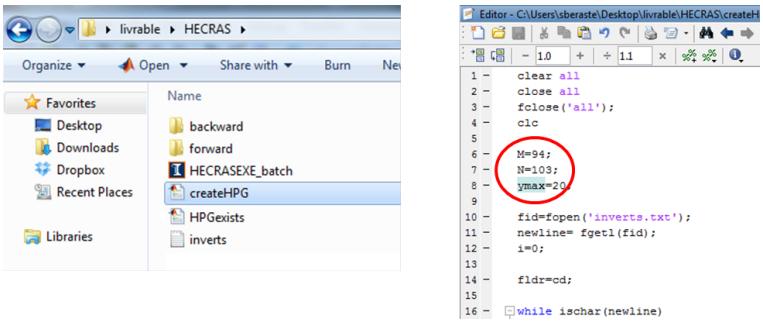


Figure A.14: code to create the HPG files

How to use the created HPGs and VPGs

A structure variable will be created and save in the matlab file *conduit* in the main folder. This variable can now be used in the SRM code.
This variable contains some properties of the conduits, and is ordered as the *conduit* field in the variable *interceptor* is ordered in the SRM code.

The created variable *conduit* has to be pasted in the folder *SetInterceptorProperties*. The Matlab file *SetConduitProperties* needs to be changed. All the instructions appear in the code as comments:

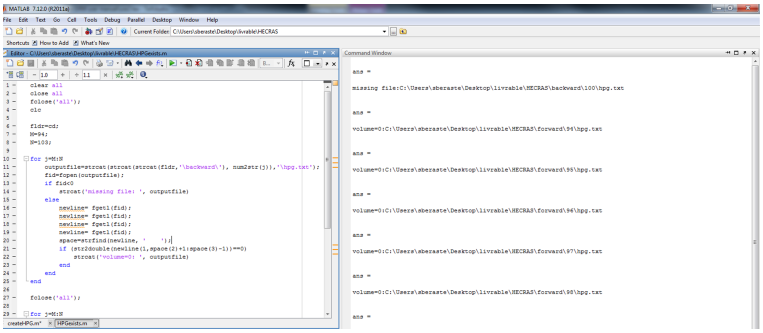


Figure A.15: How to check the files

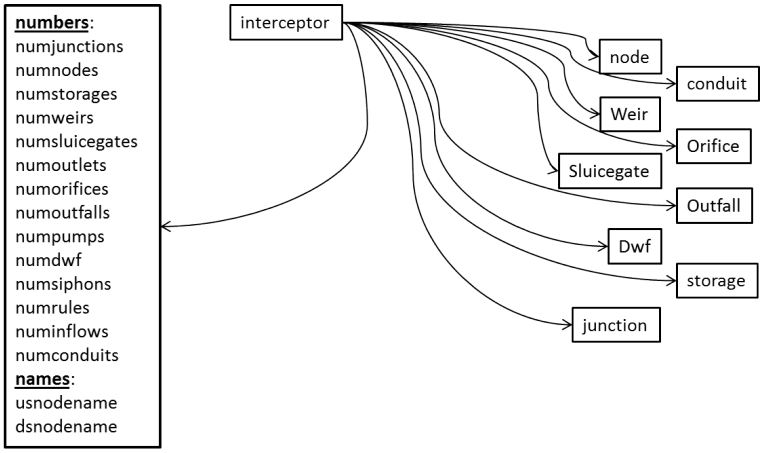


Figure A.16: the *interceptor* variable

- line 7: uncomment to load the *conduit* variable
- lines 30-78: comment all these lines
- lines 92-142: comment all these lines
- line 145: uncomment to get the HPG
- line 155: comment this line
- lines 269-270: comment two "ends"

The rest of the code is not changed. Therefore, it is necessary to set everything else as usual, it is still necessary to create a SWMM file, but the shape of the conduits are not important in this case.

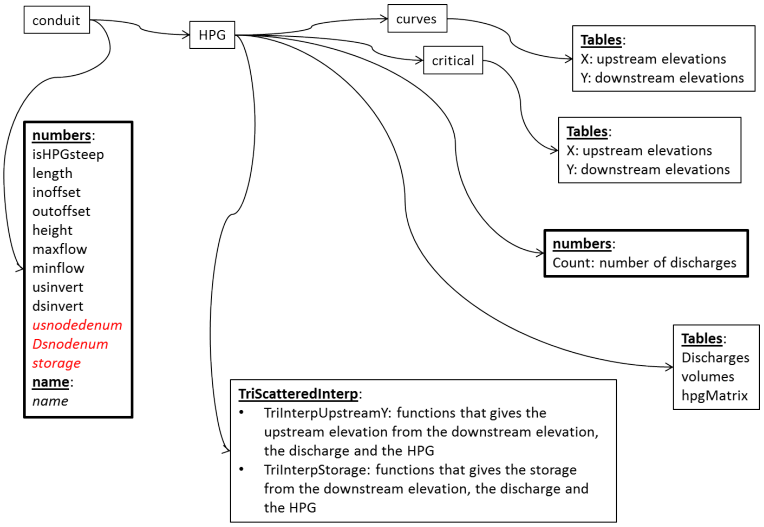


Figure A.17: the conduit *field* of the *interceptor* variable, the red fields are asbent of the conduit variable created to input the HPGs/VPGs

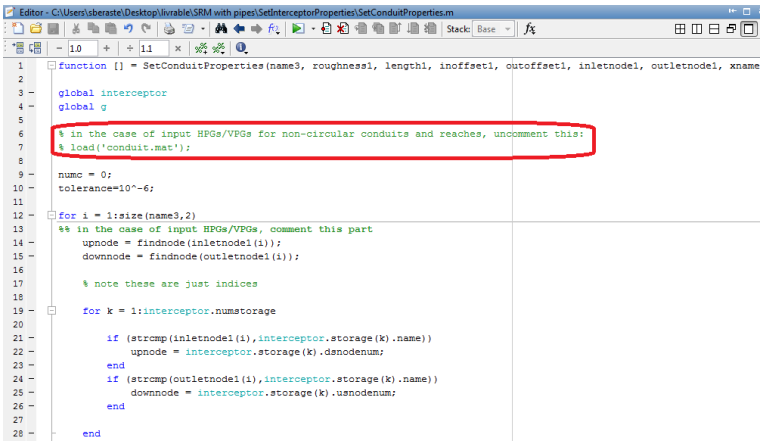

Editor - C:\Users\sberaste\Desktop\livrable\SRM with pipes\SetInterceptorProperties\SetConduitProperties.m
function [] = SetConduitProperties(name3, roughness1, length1, inoffset1, outoffset1, inletnode1, outletnode1, xname
global interceptor
global g
% in the case of input HPGs/VPGs for non-circular conduits and reaches, uncomment this:
% load('conduit.mat');
numc = 0;
tolerance=10^-6;
for i = 1:size(name3,2)
%% in the case of input HPGs/VPGs, comment this part
upnode = findnode(inletnode1(i));
downnode = findnode(outletnode1(i));
% note these are just indices
for k = 1:interceptor.numstorage
if (strcmp(inletnode1(i),interceptor.storage(k).name))
upnode = interceptor.storage(k).dsnodenum;
end
if (strcmp(outletnode1(i),interceptor.storage(k).name))
downnode = interceptor.storage(k).usnodenum;
end
end

Figure A.18: How to use the file containing HPGs/VPGs in SRM code

B Other type of boundary conditions

The equations for the boundary conditions can be changed in the files *OptimizeSection* and *OptimizeSectionSteady*. In both files, the boundary conditions are defined in the subfunctions *FindFunctions* (wich defines the functions for the Newton-Raphson method) and *FindJacobian* (wich defines the derivatives of the functions, the Jacobian, for Newton-Raphson).

```
% the boundary conditions are here
for i = 1:NumNodes

    if (strcmp(interceptor.node(i).type,'inflow'))
        eqcount = eqcount + 1;
        F(eqcount,1) = Q(i) -currentflow(i);
    end

    if (strcmp(interceptor.node(i).type,'outfall'))
        eqcount = eqcount + 1;
        F(eqcount,1) = y(i) -ycritical(i);
    end

end

end
```

Figure A.19: the boundary conditions in the code

```
% the derivatives of the boundary conditions are here
for i = 1:NumNodes

    if (strcmp(interceptor.node(i).type,'inflow'))
        eqcount = eqcount + 1;
        dFdQ(eqcount,i) = 1;
    end

    if (strcmp(interceptor.node(i).type,'outfall'))
        eqcount = eqcount + 1;
        dFdQ(eqcount,i+NumNodes) = 1;
        dFdQ(eqcount,i) = -dycrit(i);
    end

end
```

Figure A.20: the boundary conditions in the code

In the case of the functions for Newton Raphson, all the values are stored in the variable F. The size of the vector is $(1, n)$, where n is two times the number of nodes in the system (one time for the water surface elevation and one time for the discharge). The values for the boundary conditions are the last values of this vector.
In the case of the Jacobian, all the values are stored in the variable $dFdQ$. The size of the matrix is (n, n). The values for the boundary conditions are in the last lines of this matrix.

C A calculation under other assumption

As explained in the introduction, SRM is based on the quasi-steady dynamic wave assumption. In the files *OptimizeSection* and *OptimizeSectionSteady*, in the functions, this assumption can be changed, the functions for Newton-Raphson and their derivatives (the Jacobian) can be changed.